TopSolid 2013

What's New

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Contents

TopSolid'Design 2013: What's New

This document describes the improvements made to the **design** application of the **TopSolid'Design** software: **2013** version.

Installation

Prerequisites

The installation procedure proposes to install or upgrade the components listed below with the following versions:

Component	Version
Microsoft .NET Framework	2.0
Microsoft .NET Framework	4.0
Visual Studio C++ Redistributable	2005
Visual Studio C++ Redistributable	2008 SP1
Visual Studio C++ Redistributable	2010
Sentinel driver	7.5.7
Sentinel RMS Licence Manager (floating license manager)	8.5.1

Assistance

In order to provide you a better quality of service and help you to quickly solve your problems, the new **Help | Remote assistance** function allows your favorite support team to take control of your computer using the **TeamViewer** software.

For those who are using a specific remote control system, a configuration keyword allows you to define the path to another application:

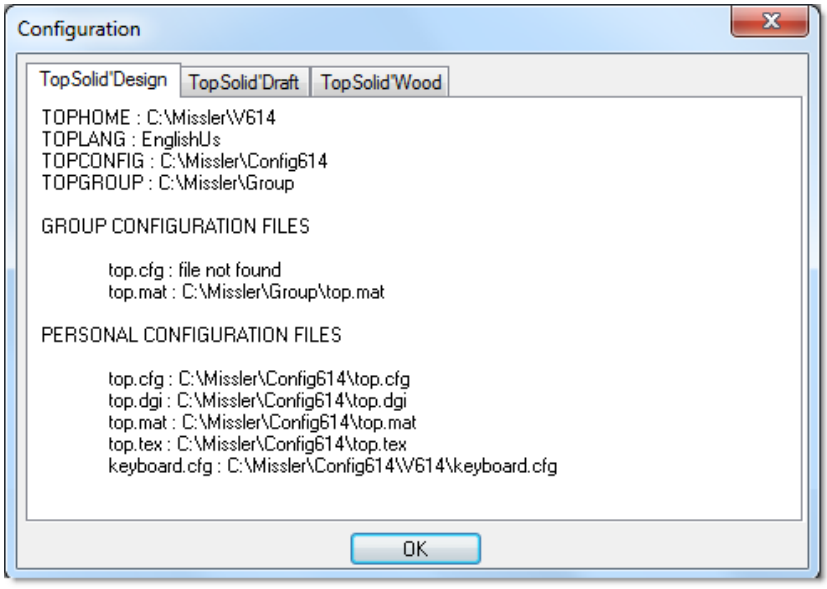
Syntax:

U_APPLI_REMOTE_ASSISTANCE_NAME <tab> **application name**

Configuration

The **Help | Configuration** function has been modified in order to display the configuration files of each application in separate tabs.

For each file, the path is displayed and if a file is not used because the application does not manage it in the *GROUP* folder for example, this information is displayed between comas at the end of the file name.

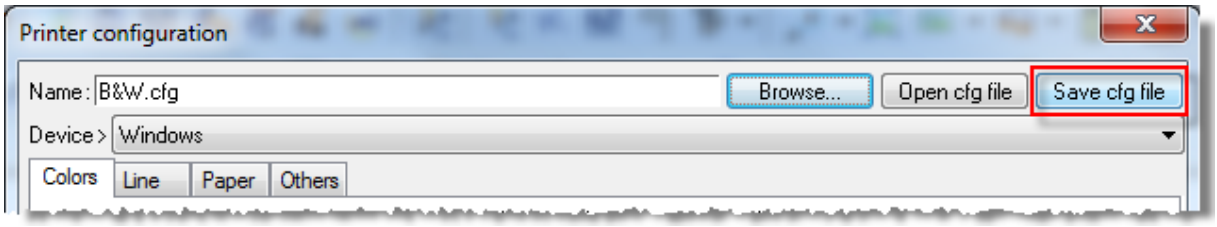


The new configuration dialog box

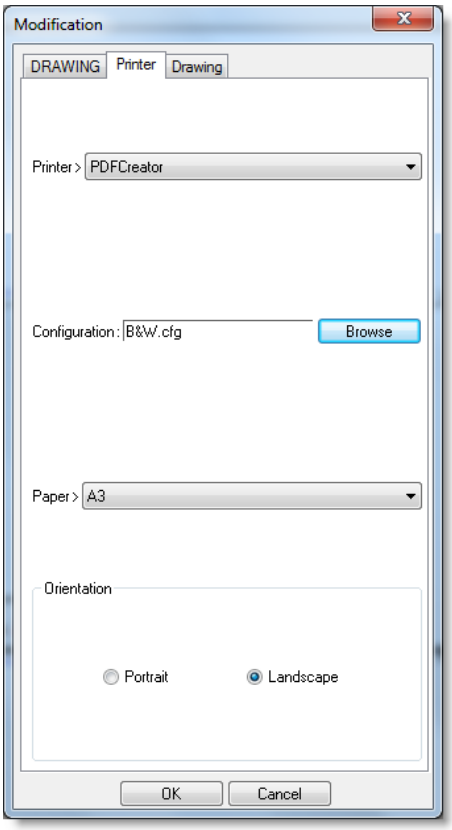
Document

Print settings in the drawing

When a document contains several drawings, the print settings can now be defined in each drawing.
First of all, the different settings (color, thickness, etc.) must be performed using the **File | Page setup** function, and then saved into a configuration file using the **Save cfg file** button.



Therefore, several configuration files can be saved according to how the drawings will be printed.
Then, when modifying the drawing, the new **Printer** tab allows you to choose the printer, the configuration file, the paper size and the document orientation.

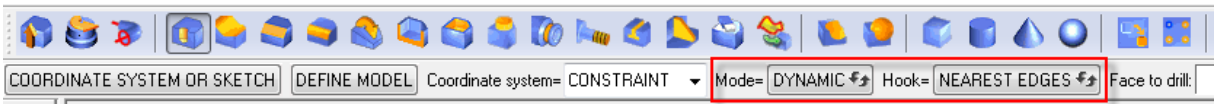


These settings can be performed in the template document; they will be used first during a simple or a multiple printing.

User interface

Dialog

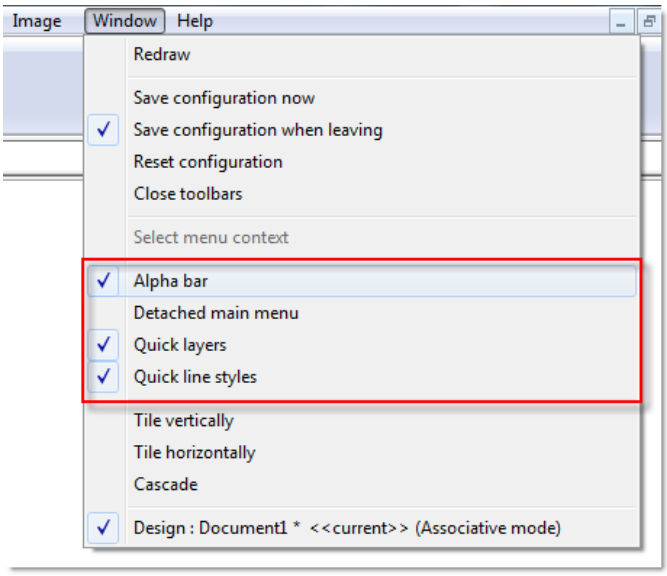
In the dialog bar, when several options are available in a mode, a small symbol made up of 2 arrows is displayed at the end of the button.



*Example of dialog bar of the **Shape | Drilling** function.*

Menus

The display of the alpha bar, the quick layer bar, the quick line style bar and the main menu is now managed as modes in the **Window** menu.

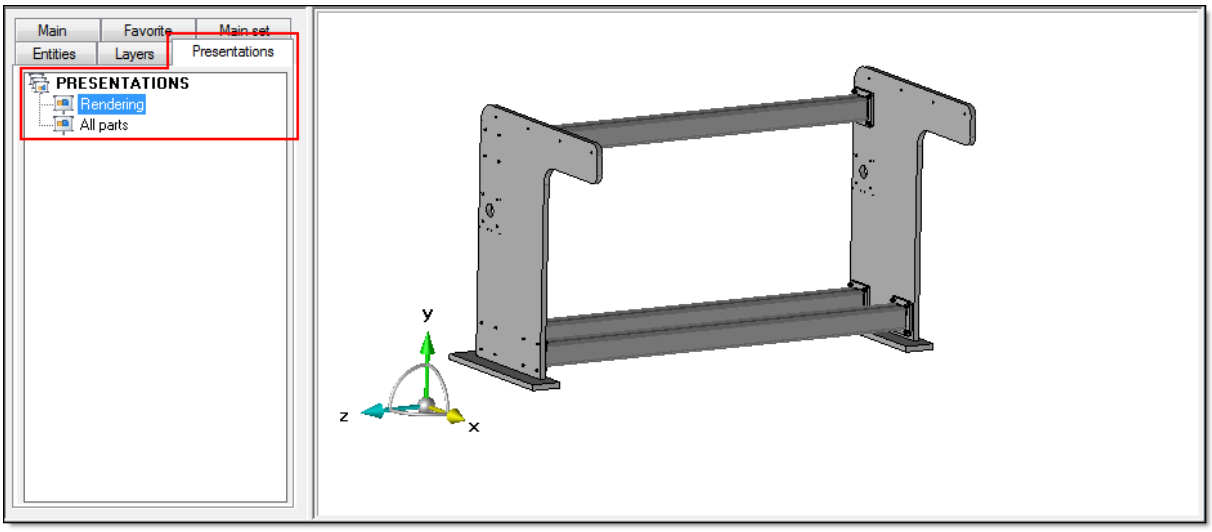


Presentation

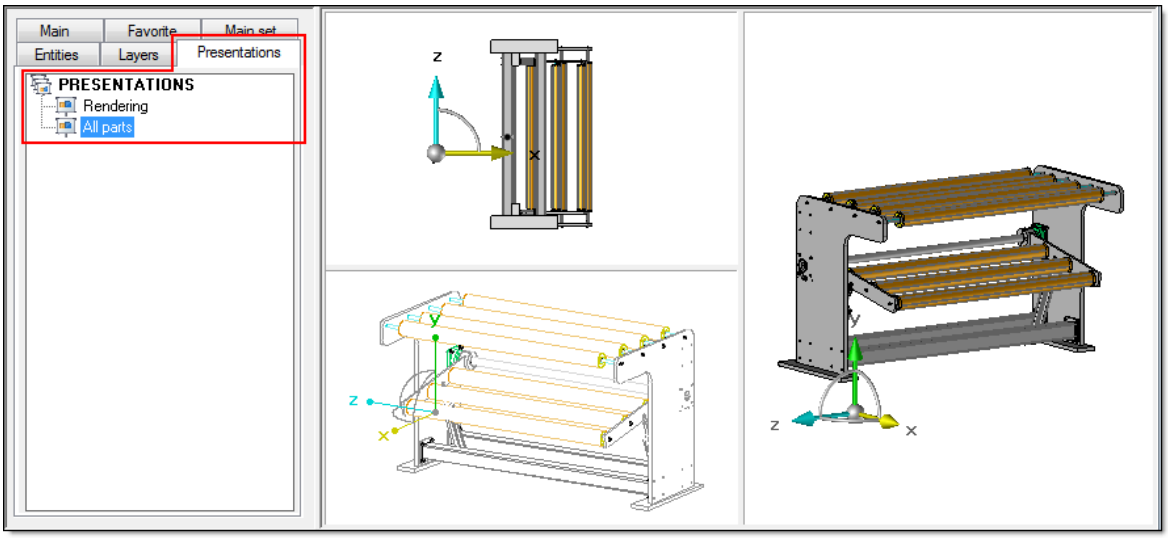
The new **Tools | Presentation** function allows you to save all the graphic data (orientation, rendering mode, zoom, multi-windows, graphical cross section, active layers, visualization mode of hidden elements) of a document in order to recall them later.

These presentations are automatically included in the new **Presentations** tab of the tree in which a contextual menu allows you to:

- add a presentation;
- rename a presentation;
- display a presentation (a presentation can be displayed by dragging and dropping it into the graphical area or by double-clicking on it);
- delete a presentation;
- redefine a presentation;
- start a slideshow from the selection of several presentations or all the presentations;
- calculate an image for each representation from the selection of several presentations or all the presentations (**TopSolid'Image** required).



Presentation in rendering mode with one view and one layer.

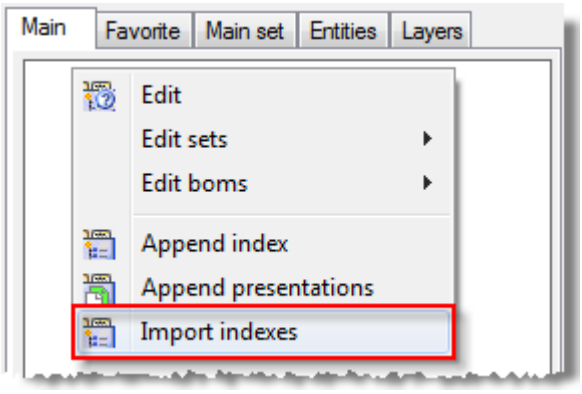


Presentation in rendering and wireframe mode with 3 views and all the layers.

Moreover, the presentations (without the graphical cross sections) are also available in **TopSolid'Viewer** if the **D_DOC_SAVE_ALL_VISUALIZATION** configuration keyword is set to 1.

Tree

The new **Import indexes** contextual command allows you to get the tabs created in another document.
If the imported tab contains a set which also exists in the current document, such as the set of lights, it is automatically reconnected.
If the imported tab contains a set which does not exist in the current document, but is created automatically by **TopSolid**, then this set is created and the tab is reconnected (e.g. drivers set, key points set, auxiliary elements set, publishings set, alternative sets are also covered).

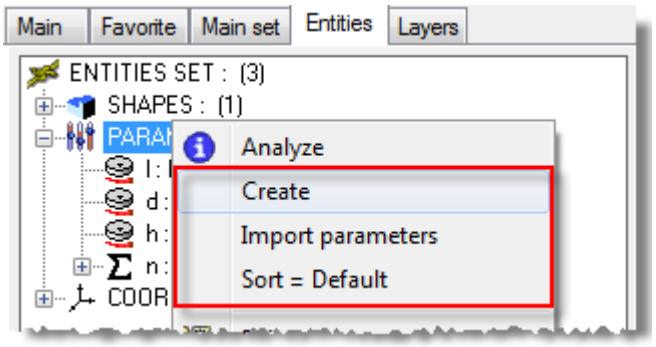


Parameter

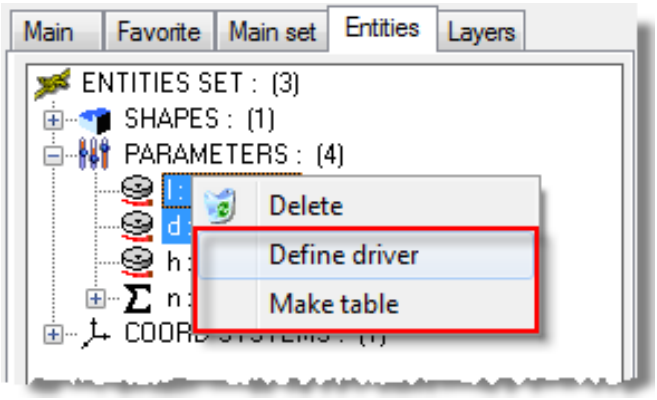
When creating or modifying a parameter, the new **PARAMETER** option now allows you to enter an expression in the minimum and/or maximum value of a parameter.

Bringing up the contextual menu on the **PARAMETERS** folder now allows you to:

- create a parameter;
- import parameters from another document;
- sort the parameters.



From a selection in the **PARAMETERS** folder of the Entities tree, you can now create a table and define parameters as drivers.



In the **Edit list** dialog box, the sorting order is now kept in the current session of TopSolid.

Edition

Repeat

Points, coordinate systems and texts can be excluded from a repetition.

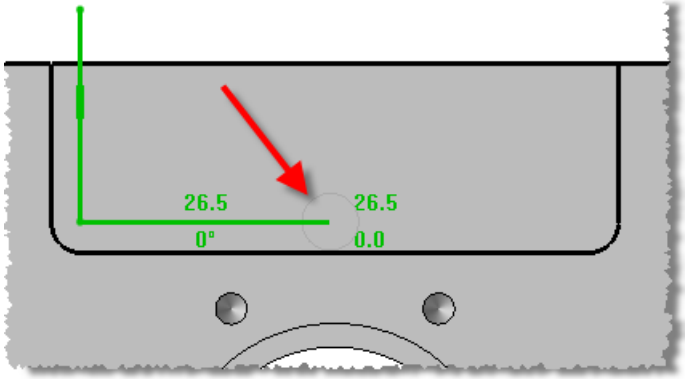
Name

When naming an element from the tree, it is now possible to add a designation.

Curve

Contour with machining circle

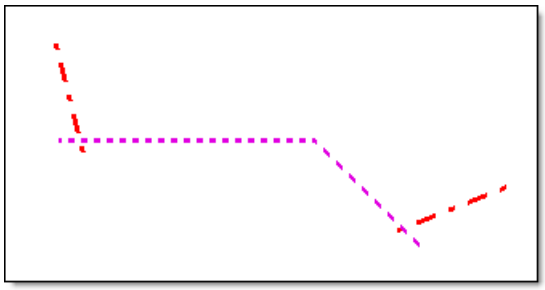
For example, when it is required to manually draw the tool path inside a pocket, the **WITH MACHINING CIRCLE** option, available in the advanced options, allows you to display the tool bulk under the cursor.



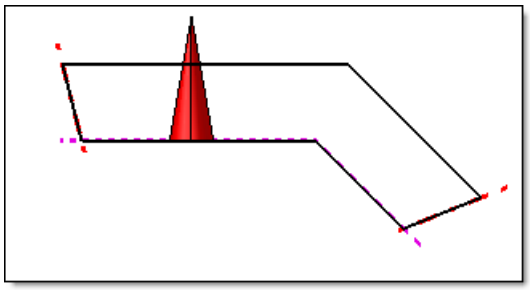
Example of contour with machining circle.

Thickened curve

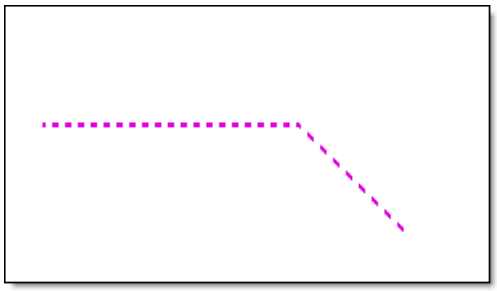
The **TRIM BY LINES** and **MITRE CUT** options have been added to the drop-down list for choosing the type of end. They enable you to trim a thickened curve either by lines, or by mitre cut.

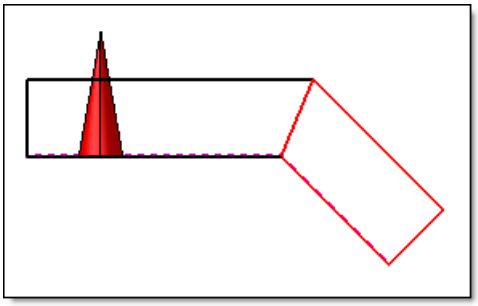


In magenta, the curve to thicken and in red, the trimming curves.



In black, the thickened curve.





On the left, the curve to thicken and on the right, the selection of the 1st segment allows you to get 2 thickened curves (in black and red).

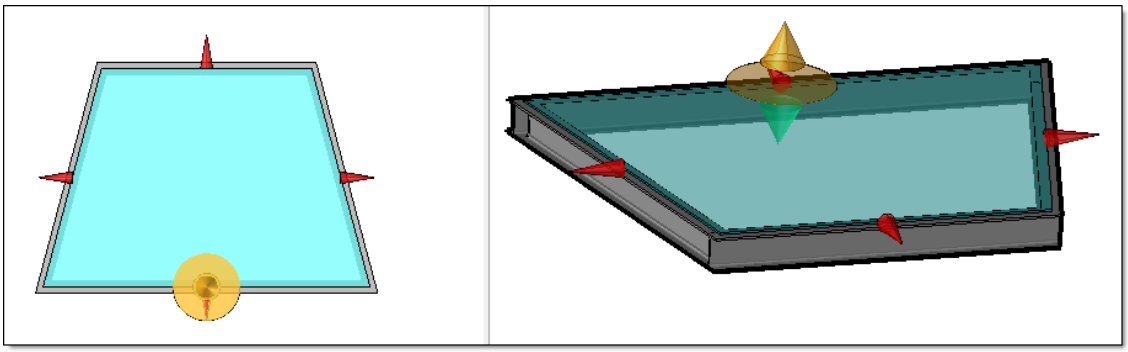
Smoothing

Now, the number of given point is associative; it can be modified using the **Modify element** function or from the tree.

Shape

Free constrained block

The new **Allow non-parallel faces** option allows you to create a constrained block on 4 planes whatever their orientations.



Example of constrained block creation (in blue) on a trapezoidal structure made up of profiles.

Deactivation of a group of operations

Now, a group of operations can be deactivated with a condition.
By this way, the deactivation condition is set to the group, the operations inherit this condition and all the new operations included in the group too.
When an operation is extracted from the group, its deactivation condition is kept.

Trimming

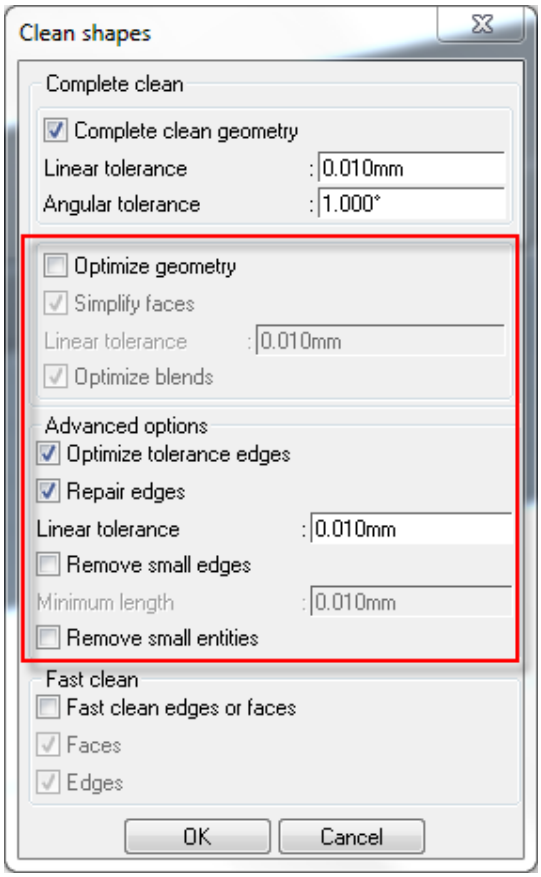
The dialog of the trimming by an imprint curve has been modified; now, the extension length is only asked when the trimming curve is open.

Coating

Now a coating operation can be modified from the tree using the **Modify** contextual function.

Cleaning

New options have been added to the **Shape | Manage | Clean geometry** function.



*The new options of the **Clean geometry** function.*

The **Repair edges** option also allows you to make a repairing on edges of faces. You can specify the edge tolerance to follow and it is recommended to enter a tolerance greater or equal to the global cleaning tolerance.

The **Optimize blend** option allows you to find the "fillet" information on surface according to the Parasolid modeler definition. In order to get a lighter model, the optimization of the shape after simplification is recommended by Parasolid.

The **Simplify faces** option allows you to make a simplification in addition to the repairing.

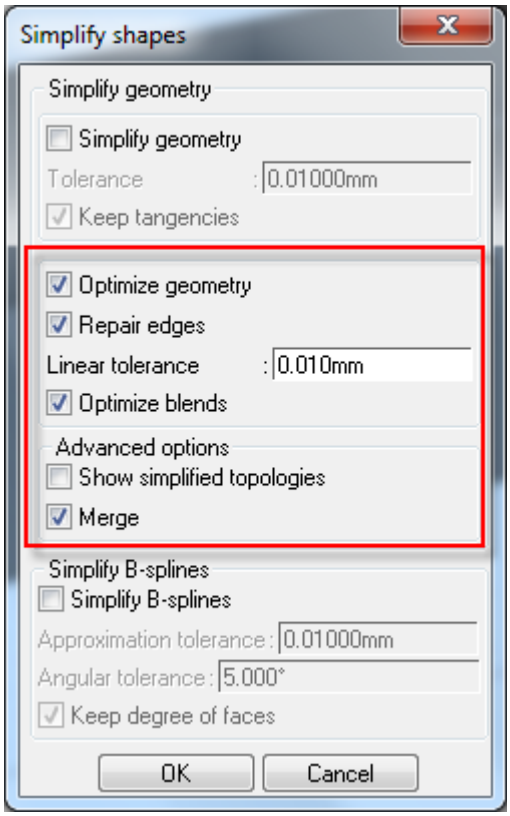
The **Remove small edges** option allows you to delete edges whose length is less than the specified value. Then, the surface is rebuilt on the remaining edges.

The **Remove small entities** option allows you to delete the micro-surfaces which could be ignored by considering only the neighboring surfaces to design the shape. The deleted topologies are:

- small edges and small faces;
- peaks;
- thin faces;
- notches.

Simplification

New options have been added to the **Shape | Manage | Simplify geometry** command.

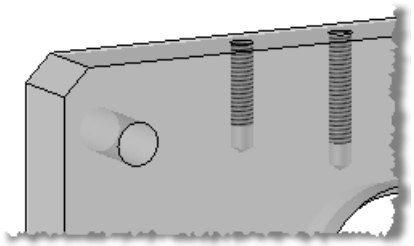


*The new options of the **Simplify geometry** function.*

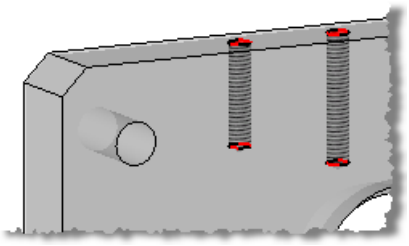
The **Keep tangencies** option allows you to keep a continuity of tangency between simplified faces.

The **Repair edges** option is the same than the one available in the **Clean geometry** function.

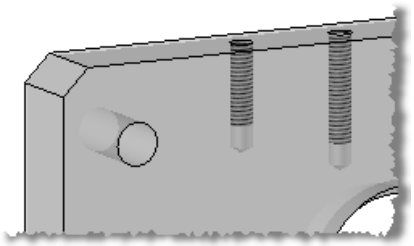
The **Merge** option allows you to merge redundant geometries such as the hole face and the threading face of a tapping hole. In the previous versions, redundant geometries were automatically merged. This means that in a blind tapping hole, the threading face was merged with the hole face and was extended up to the bottom of the hole.



Shape to simplify



Result of the simplification with merge: the threading face is merged with the hole face.



Result of the simplification without merge: the threading faces are identical to the original shape.

Superposed surfaces

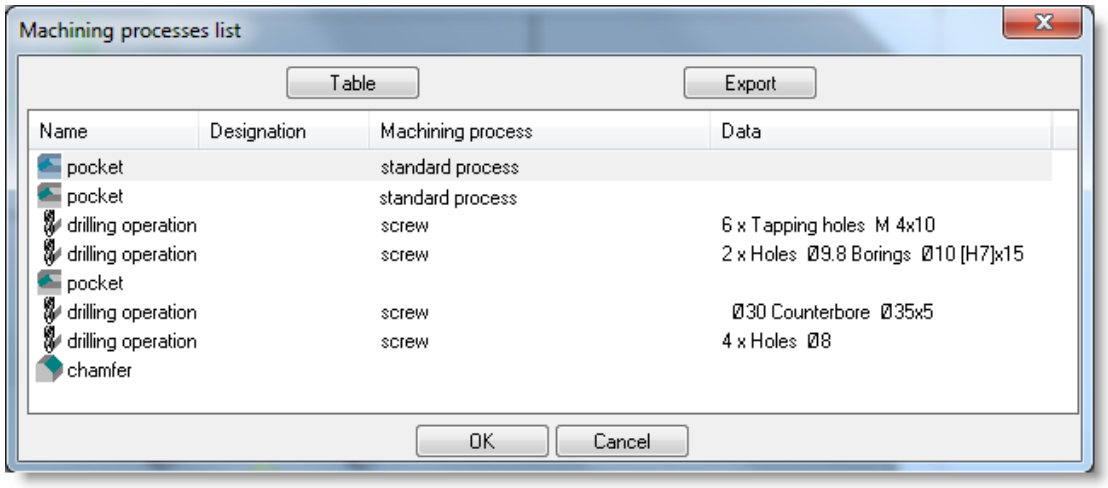
Now, the superposed surfaces can be deleted from the tree.

Checking in multi-core mode

The check geometry functions such as the **Shape | Manage | Check geometry** function have been optimized in order to automatically use the available processors. This allows you to save up to 30% of time.

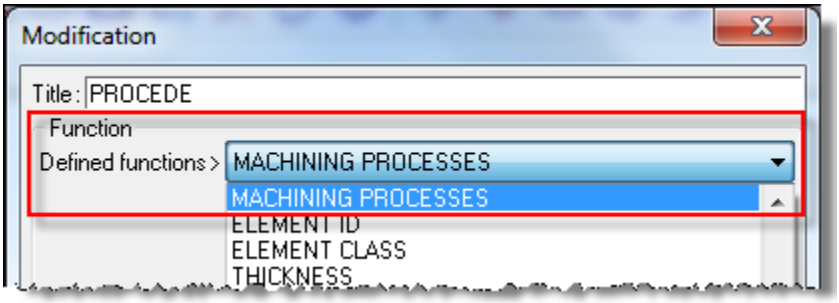
Machining process checking

The new **Shape | Manage | Check machining processes** function allows you to display and modify all the machining processes defined in the operations of a part.
Two buttons are also available; they allow you to export the list into a text or an **Excel** document and create an associative table in the document.



Example of a machining process list for a part.

Moreover, the machining process list of a part can be displayed in a BOM column using the **MACHINING PROCESSES** function.

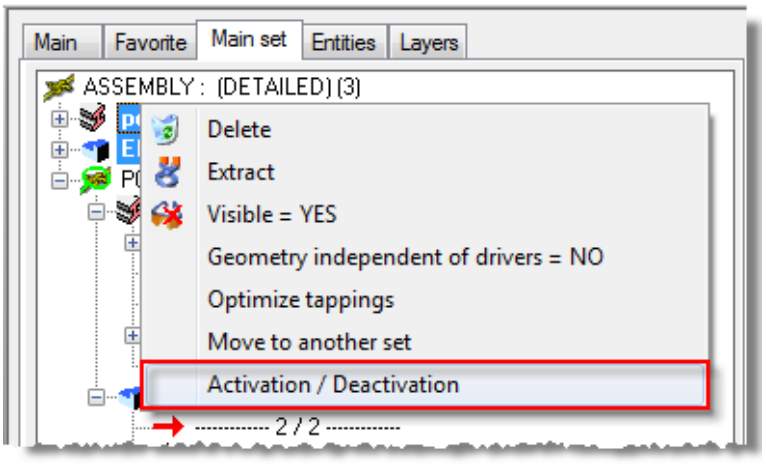


Example of a BOM template displaying machining processes.

Assembly

Deactivation

The deactivation of a part or a component with or without condition now can be done from a selection in the tree.



Alternative set

Now, in-place assemblies can be included into alternative sets as well as the main set.

In place assembly

The multi-selection icon  is now available when including a part in an in-place assembly.

Management of slave part modifications

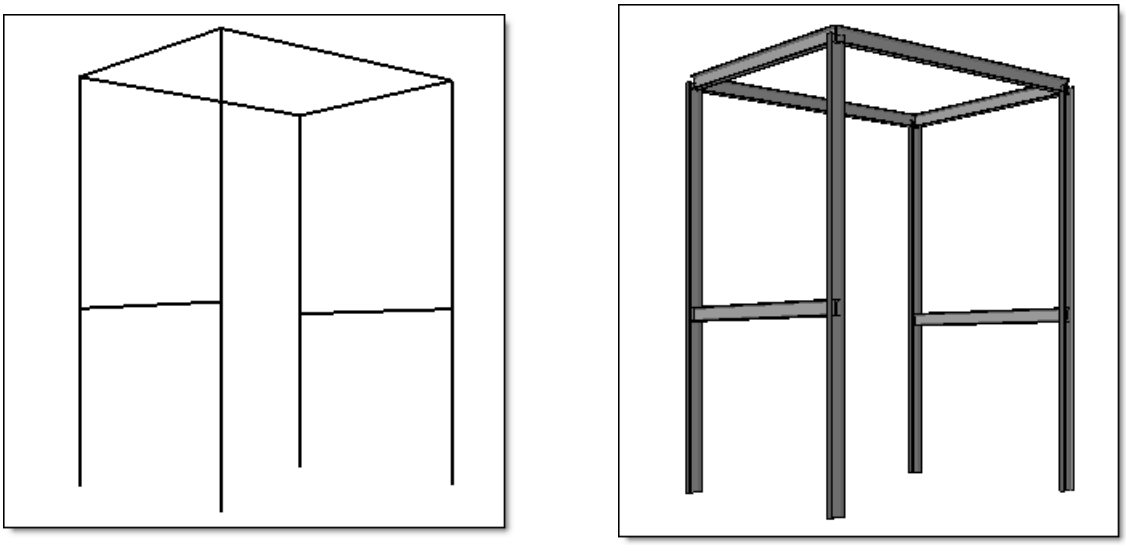
In a document containing a slave part, when modifying the part's characteristics, only the **Cutting-up**, **Stock** and **Machining** tabs are available and in the **Cutting-up** tab, only the over dimensions can be modified.
For example, when the over dimensions of a slave part are modified, the properties are forced. From the tree, the **Initial characteristics** contextual function allows you to get the characteristics of the master part.

Components

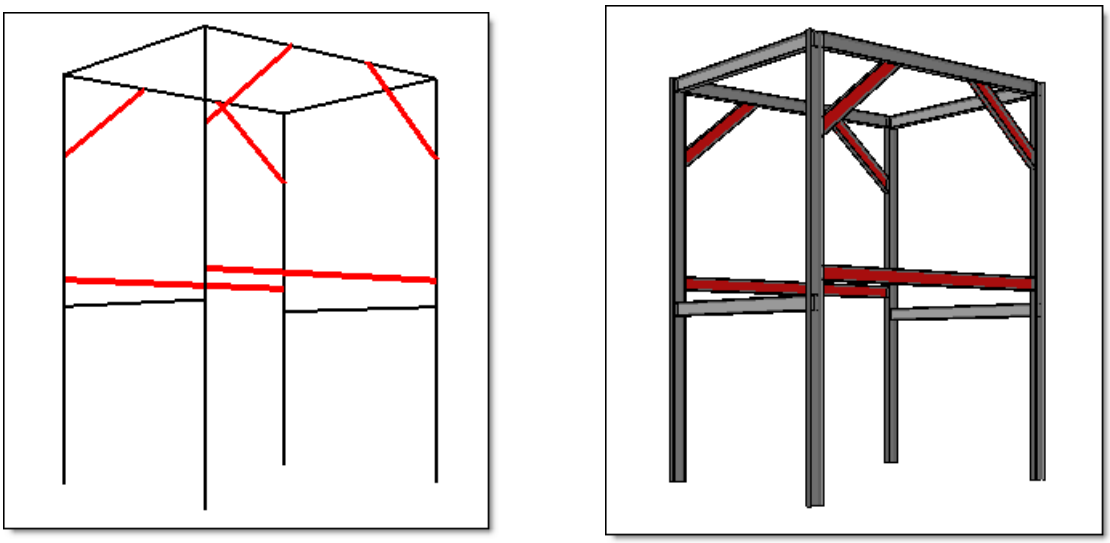
Profiles on curves

When including a profile component, the new **ON CURVES** option allows you to create the profiles on each linear segment of a contour or a sketch with or without mitre cut (mitre cut, covering/covered mode,…).
The profiles thus created are grouped into an entity named **Multiple component process**, and you can then:

- modify one or all the profiles;
- add or remove contours;
- modify the cut type;
- modify the key point and the rotation angle.



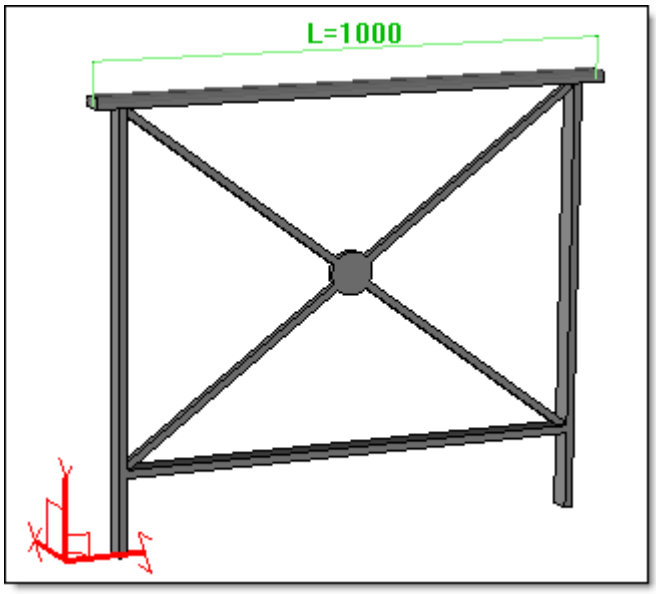
*Example of a wireframe structure on the left and the structure with profiles on the right; all the profiles are included at the same time thanks to the **ON CURVE** option.*



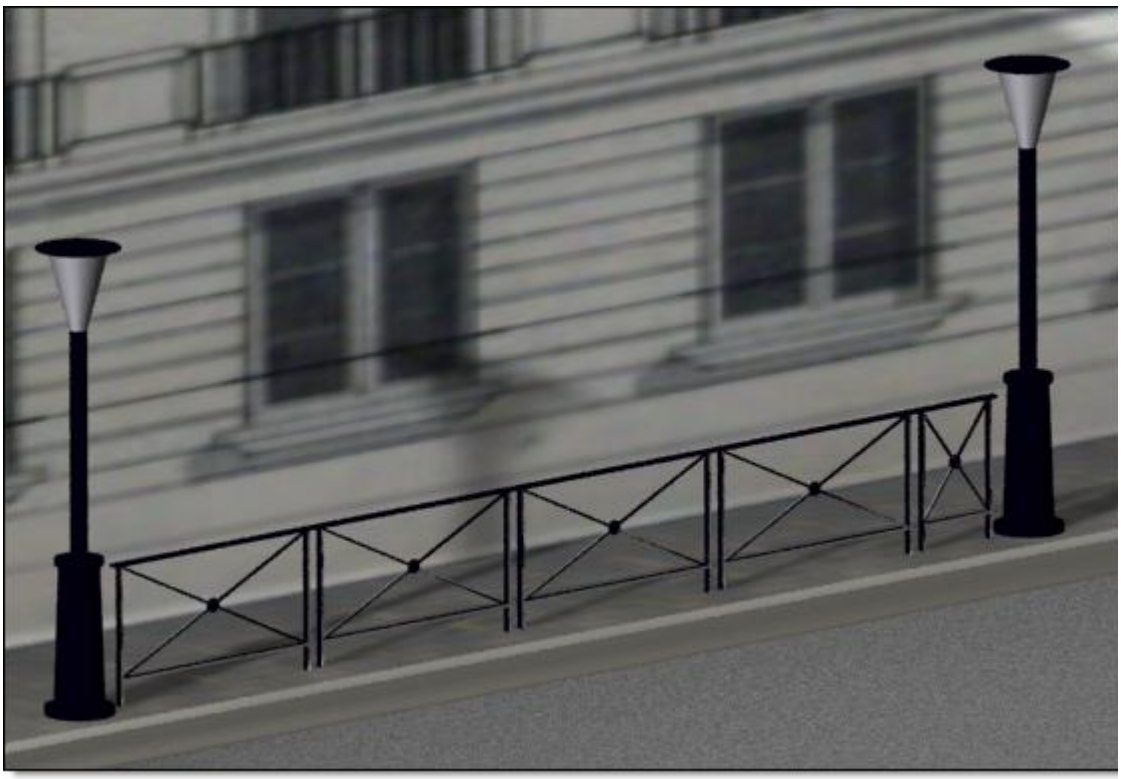
The structure is completed with new curves (on the left) and new profiles are automatically added (on the right).

Distribution

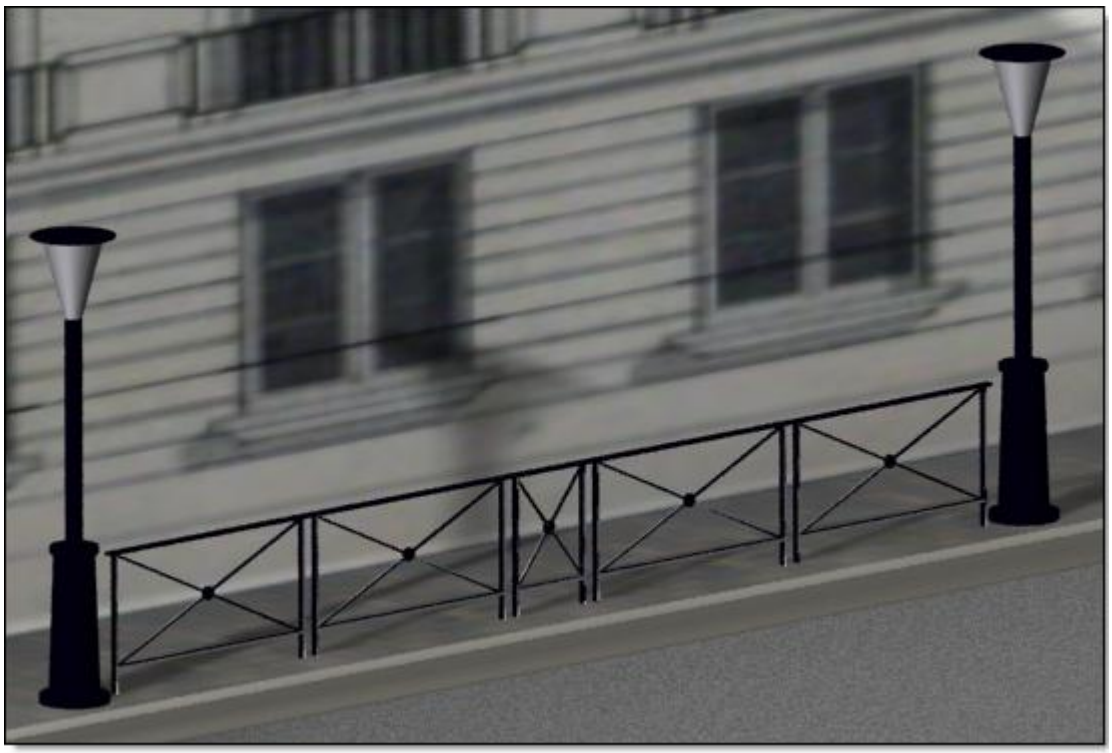
The distribution allows you to distribute components between two stops by modifying the length of one instance automatically such as a rail guard between two walls.
It is implementing by the new **Assembly | Define component | Define distribution positioning** function which allows you to define the positioning coordinate system and the driver parameter to modify.
Then, during the inclusion, the new **DISTRIBUTION** option allows you to include the component between two planes either by automatically modifying the driver's value of the last instance, or by leaving an empty space.
The component whose dimensions have been modified can be positioned either at the end of the distribution (**Sequential** mode), or in the middle (**Alternated** mode).
The distribution can be modified globally (modification of parameters, replacement of component,…) or locally on one component only (sections).



Example of distribution positioning.
In red, the positioning coordinate system whose Z+ axis is oriented towards the distribution direction and the driver parameter named L which manages the rail guard length.



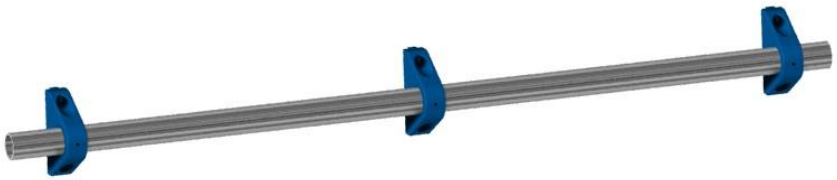
Example of distribution between two stakes; the latest instance is automatically modified in order to fill the empty space.



Example of distribution between two stakes; the midst instance is automatically modified in order to fill the empty space.

Driver propagation

A linear, linear constraint and circular propagation can now be defined as a driver, which allows you for example to shift one or several occurrences.
Once the propagation has been defined, an offset parameter is set on each occurrence and you can shift an occurrence along the propagation direction with a double click or with the **Shift instance** contextual function. An offset in the opposite direction can be done by entering a negative value.
When a propagation is defined as a driver in the template document of a component, the offset of the instances can be done in the assembly document using the **OFFSET INSTANCE** of the **Modify element** function.



Example of component, the linear propagation of the blue sub-component has been defined as driver.

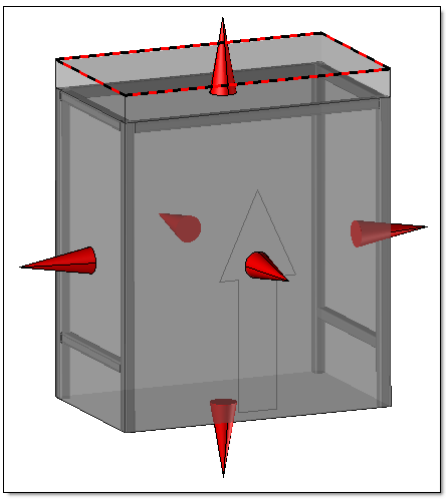
In the assembly, on instance is not properly positioned.

Then, it is possible to shift one instance of the driver propagation

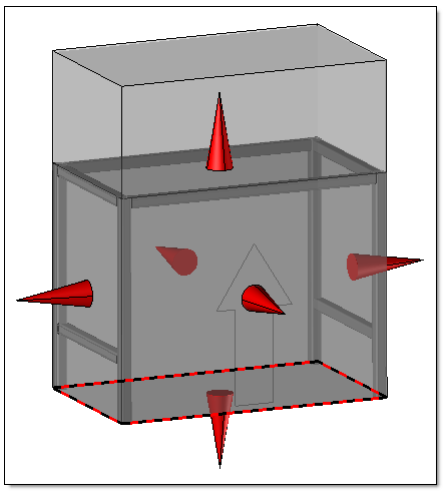
Driver block

The dialog for the inclusion of a component containing a driver block has been modified. The different modes are now displayed in a drop-down list and the **NEW CONTOUR** mode allows you to create the housing on the fly.

When the housing has been selected, the arrows allow you to choose a repositioning plane and define an offset according to the plane and/or a length.



Inclusion with offset



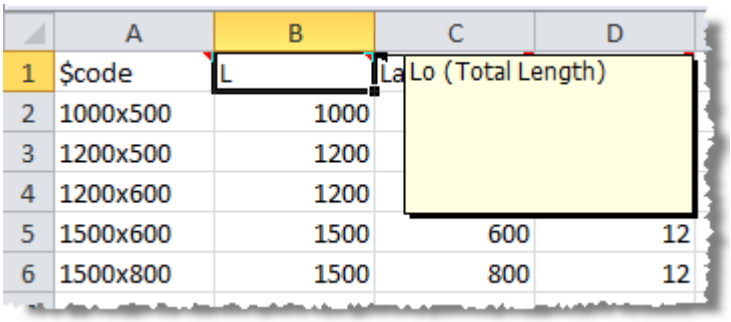
Inclusion with length

Multi driver block

Moreover, when the component contains drivers, the multiple inclusion mode allows you to include several components in several housings with the same driver parameter values.

Excel catalog

When creating a catalog using the **Assembly | Components | Edit catalog header** function, the parameter designations are now exported as comments in the Excel cells.



	A	B	C	D
1	$code	L	La	Lo (Total Length)
2	1000x500	1000		
3	1200x500	1200		
4	1200x600	1200		
5	1500x600	1500	600	12
6	1500x800	1500	800	12

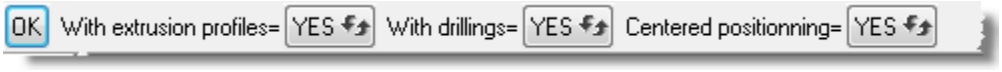
When the document contains a big amount of parameters, the display of the designation will be helpful.

Boolean drivers

When a component contains Boolean drivers defined one after another in the driver set, the new **D_COMPO_ASKPARS_BOOLEANS_SIMULT** configuration keyword allows you to group them by 2 or 3 during the inclusion of the component.

Syntax:
D_COMPO_ASKPARS_BOOLEANS_SIMULT <tab> **nb** (nb can be equal to 2 or 3)
Example:
D_COMPO_ASKPARS_BOOLEANS_SIMULT **3**

Result:



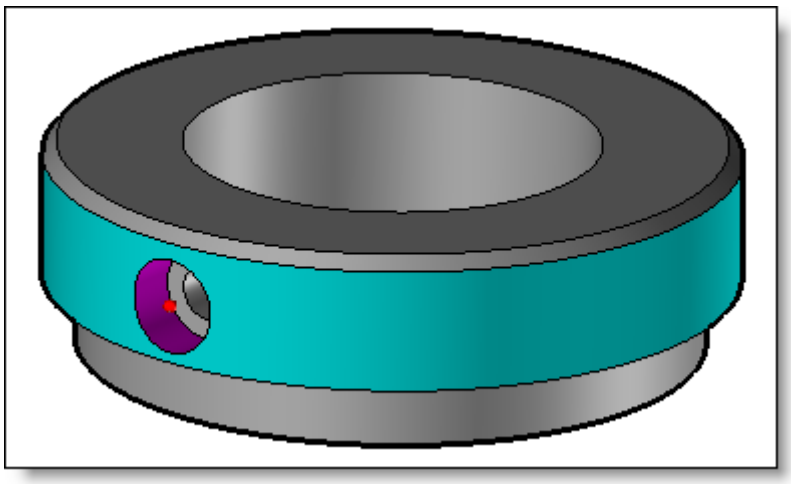
Modification of parameters

When modifying the parameters of a component, the **PARAMETERS** option offers two new modes which allow you to display all the drivers in a window similar to the **Parameter | Edit list** window and to modify them.
The **CONFIGURE SINGLE** mode allows you to modify the selected component only, whereas the **CONFIGURE MULTIPLE** mode allows you to modify the parameters of several components at the same time.
When different components are selected, only the drivers with the same name are displayed.

Tools

Axis-curve/Plane-face intersection point

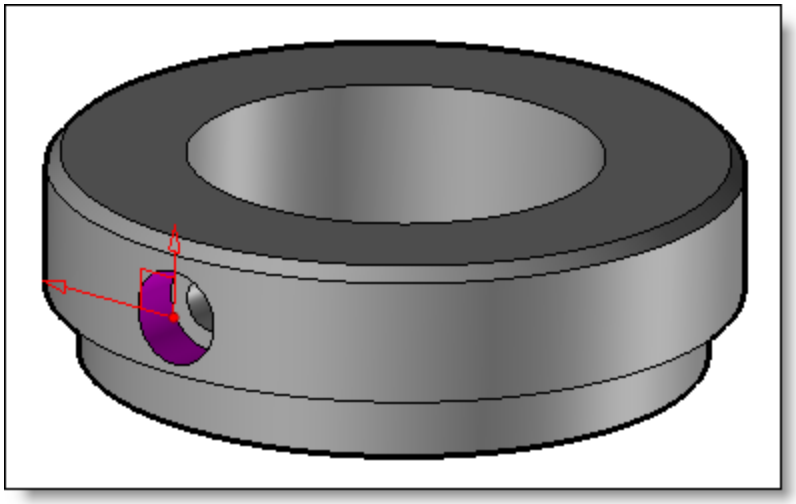
The **Trimmed face= NO/YES** option has been added to the **AXIS-FACE** mode.
With basic geometry such as imported geometry, it allows you to quickly recreate the intersection point between the cylindrical face of a hole and its reference face.



In magenta: the drilling axis, in blue: the intersected face, in red; the intersection point (reference of the drilling).

Coordinate system on axis and point

In addition, the new  **Coordinate system on axis and point** allows you to quickly recreate the coordinate system between the axis of the cylindrical face and its reference face.



With imported geometry (basic shape), the reference coordinate of a hole can be quickly created.

Coordinate systems

The relative and bisector coordinate systems are now available in the coordinate system icon bar. By this way, they can be created on the fly without ending the current function.

Attributes

Updating materials from textures

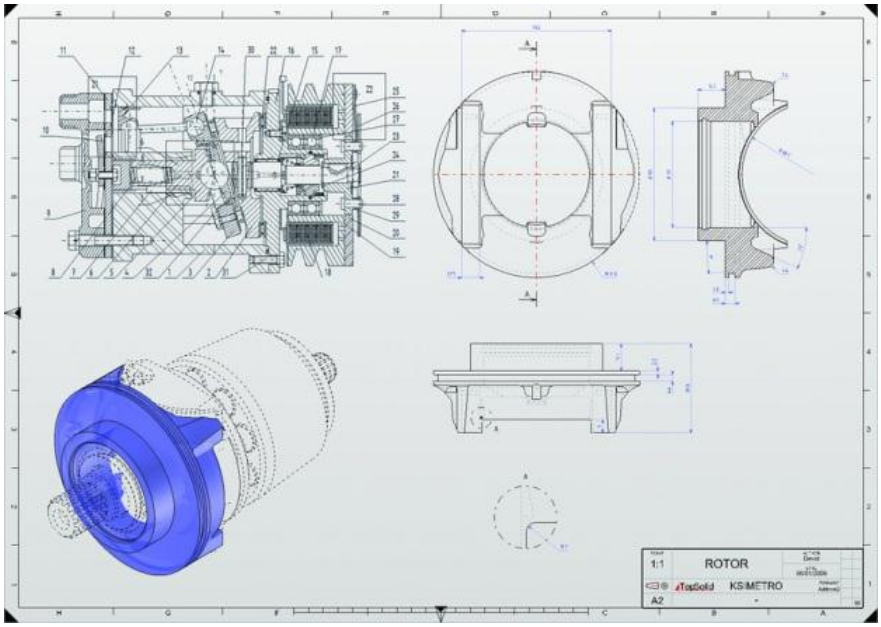
When the materials have been created from textures and then new textures are added (in a new or an existing family), these new textures are now taken into account when the function for creating materials from textures is run again.

Analysis

Collisions

When the collision analysis is performed using a multi-body shape, the message now displays the number of colliding bodies instead of the number of colliding shapes.

TopSolid'Draft 2013: What's New

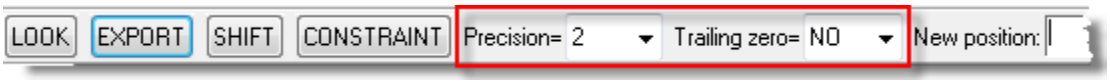


This document describes the improvements made to the **draft** application of the **TopSolid'Draft** software: **2013** version.

Dimension/Detailing

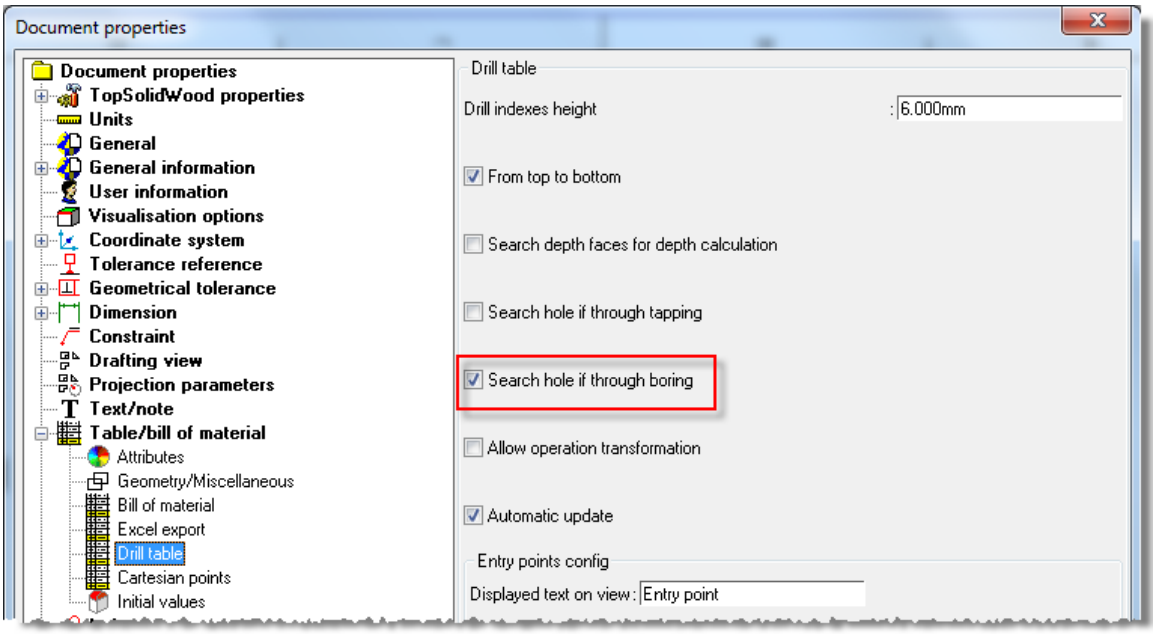
Dimension

When modifying a dimension, the first dialog now allows you to modify the number of digits and display the trailing zero.



Drill table

In the **Drill table** section of the **File | Properties** function, the new **Search hole if through boring** option allows you to distinguish the hole and the boring operations of a hole+boring drill in two separate lines.



The setting of the option in the document properties.

KEY	INDEX	SYMBOL	TYPE	COORDINATES	ZDP	DIAMETER	TOL_DIAMETER
RK	1		Hole	-55.000 * -30.000	0.00	8.00	8.00
RK	2		Hole	-55.000 * 30.000	0.00	8.00	8.00
RK	3		Tapping	-22.000 * 0.000	0.00	4.00	4.00
-	-	-	Hole	-	0.00	3.30	3.30
RK	4		Boring	-20.000 * 35.000	-5.00	10.00	10.00 H7
-	-	-	Hole	-	-5.00	9.80	9.80
RK	5		Tapping	-11.000 * -19.053	0.00	4.00	4.00

Result shown in a drill table: the hole+boring drill is displayed in two separate lines.

This setting is also available in the **Drill table** section of the **Tools | Options** function.

Moreover, the two new options **THROUGH DRILLS ONLY** and **NOT THROUGH DRILLS ONLY** allow you to create a drill table containing through holes or blind holes only.

Title block

Two new properties are now available:

- The **DAY DATE** allows you to include the printing date in a cell of a title block. This date uses the system date and is automatically updated when loading the document.
- The **LAST INDEX DATE** allows you to include the date of the last revision index and is automatically updated when a new revision is added.

Translators

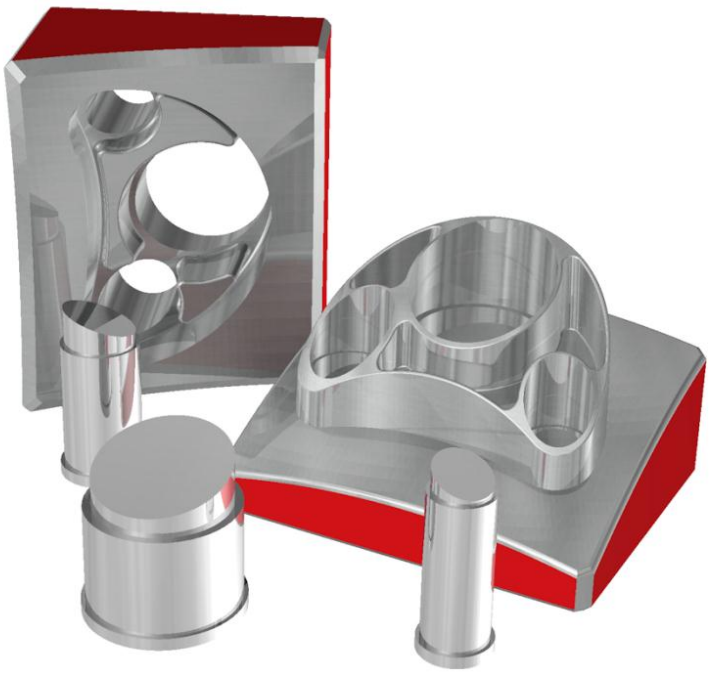
SolidWorks Spatial

SolidWorks configurations are now supported during the import.

The translators have been updated in order to support the following formats:

Translator	**Supported version**
Import	
AutoCAD	2012
Acis	R23
Inventor	2013
Catia V5	R6 – R22
	V5-6 R2012
Parasolid	V25.1
Pro/Engineer (Datakit)	2000i
	Creo 1.0
Pro/Engineer (Spatial)	16 – WildFire5
	Creo 2.0
Google Sketchup	8
SolidWorks (Spatial)	98 - 2013
Unigraphics (Datakit)	NX8
Unigraphics (Spatial)	NX8
Export	
AutoCAD	2012
Acis	R23
Catia V5	R6 – R22
	V5-6 R2012
Parasolid	V25.1

TopSolid'Mold 2013: What's New

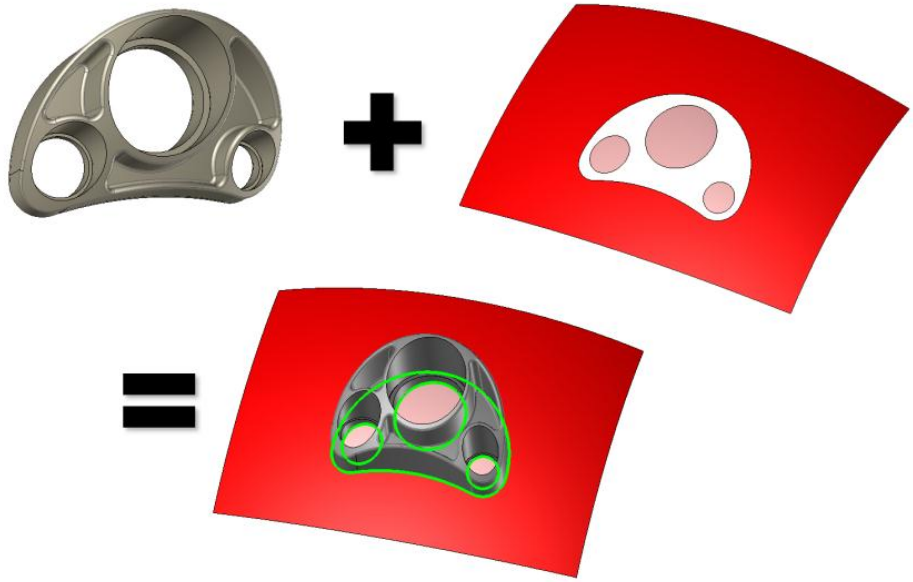


This document describes the improvements made to the **TopSolid'Mold** software: **2013** version.

Parting elements

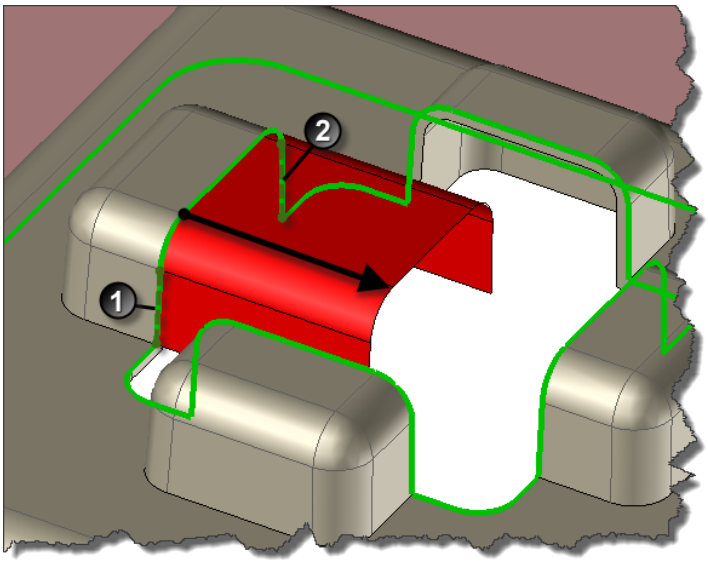
Parting lines from surfaces


The new **Parting lines from surfaces** function allows you to assign some surfaces defined in another document (TopSolid'Design creation and/or surface import from an external CAD system) in the current document. Once done, TopSolid'Mold will automatically find the parting lines related to these geometries. Of course, it is possible to reuse the shrinkage factor applied to the part to be injected, defined during its insertion. The parting set management is also done automatically.



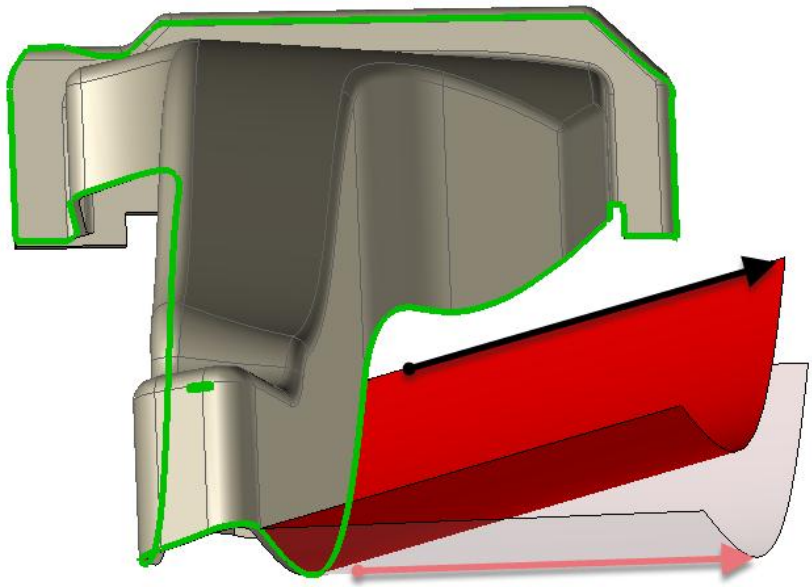
Internal extruded surface


It is now possible to define an **internal extruded parting surface** by selecting an edge path between a start edge and an end edge, as it is already possible for the external extruded parting surface.



Extruded direction

Now you can **extrude** a parting surface according a TANGENT direction, as it is possible in TopSolid'Design. This mode is available directly when creating the surface, in addition of the existing directions related to the current coordinate system.



Sewing tolerance

The **sewing tolerance**, used by TopSolid'Mold during the sewing process between parting shells and parting surfaces, is now available to the end user. You can set the default value in **Tools | Options | Blocks | Blocks sewing tolerance**. This setting can be edited and modified when creating the blocks, in the **advanced options** >>.

Movement

Guiding rail processes

After inserting guiding rails in TopSolid'Mold, it is now possible to define directly their process length. You can enter a value, or indicate on screen the expected length, as you do for the slides.

Stroke test

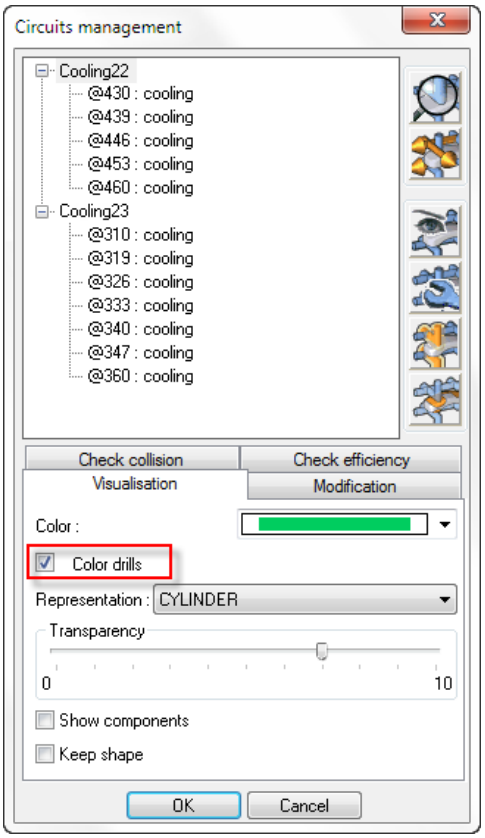
It is now possible to test the global stroke of a block assigned to a slide. This test is also available for a slide created on frame. In this way, the checking is more efficient and reliable.

Cooling system

3D drilling colorization

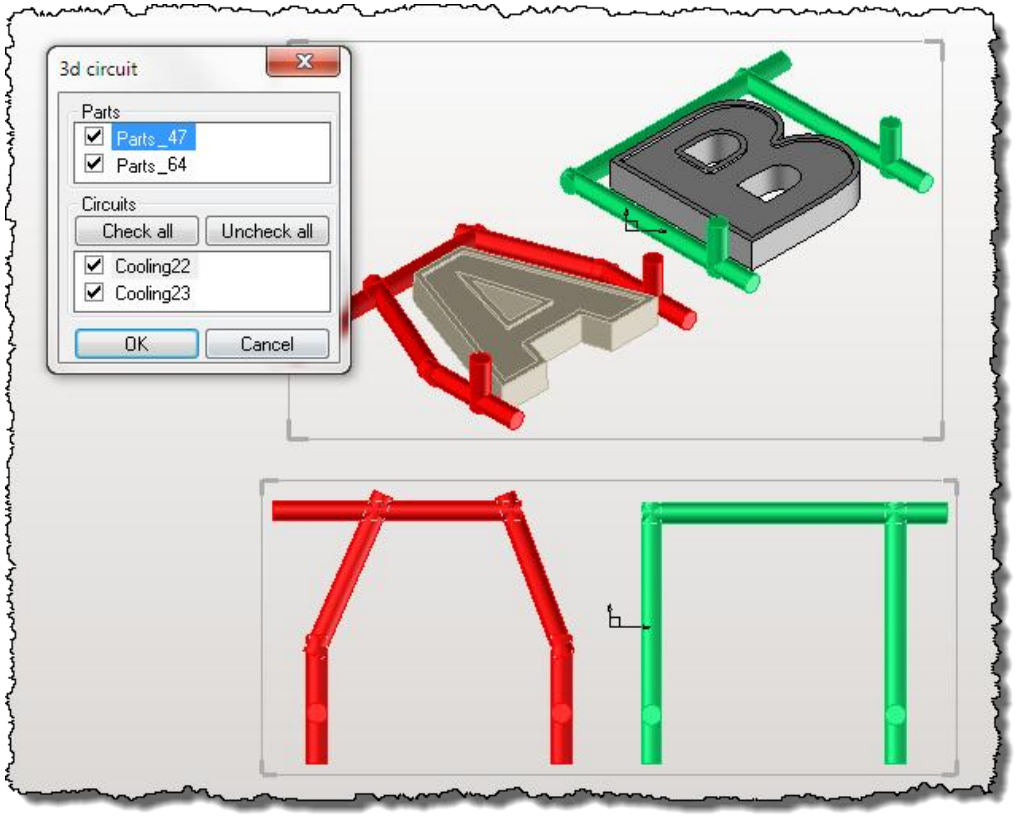
You can now color the cooling drillings in the 3D design. When renaming the circuit, a new option allows you to color or not the created circuit. The default color can be set in **Tools | Options | Cooling and Runner | Regulation Property**. The color is applied directly on the operations; in this way, you can export your colored part, for example.

This color can be modified in the **Circuits Management** window. An option allows you to color drillings, during the computation of their visualization.



Cooling circuit management in 2D

A **management tool** is now available when creating a 3D cooling system view. It is easier to select the cooling circuit(s) to project, with their associated parts, if you need them. You can find this improvement in TopSolid'Draft, from the **Mold | Draw regulation | 3D Circuit** function.

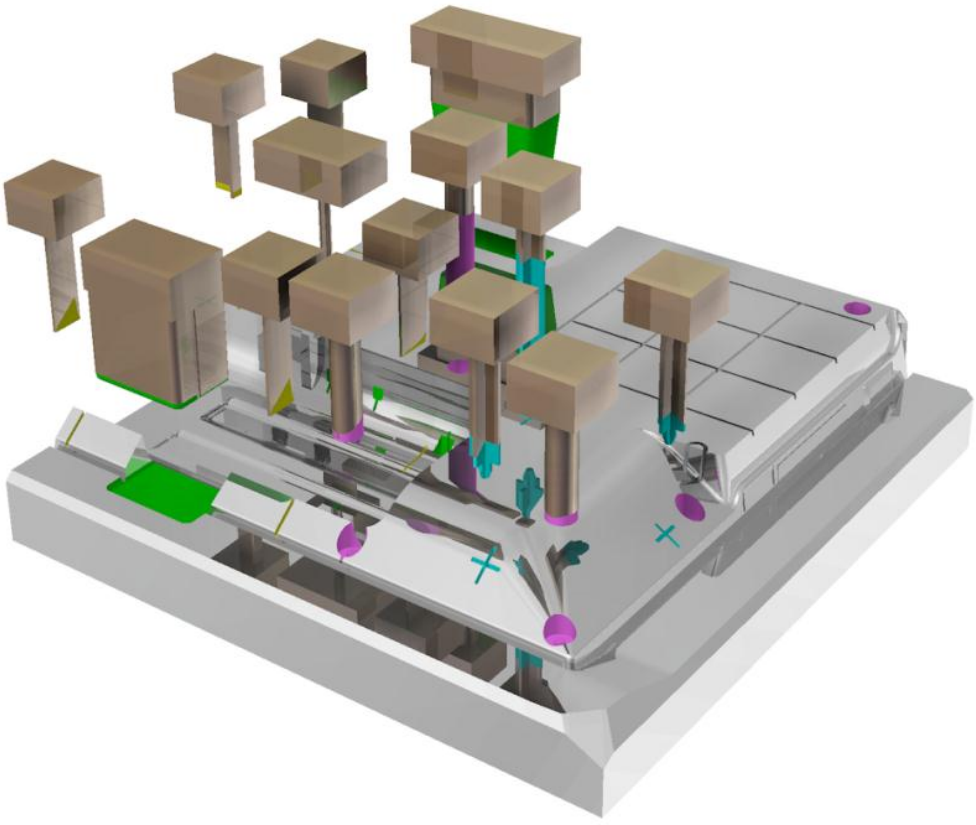


Management

Drilling modification

 It was possible to modify all the drillings of an element, if those drillings got the same type. Now, a new option has been added: you can filter the drillings according to the dimension of a reference drill. You can set this new way to modify in **Management | Modify Drills | Equivalent Diameter**.

TopSolid'Electrode 2013: What's New



This document describes the improvements made to the **TopSolid'Electrode** software: **2013** version.

Creation

Computation of the applied spark gap

Sometimes, the electrode shell offset failed due to an internal kernel error. It was possible to create the electrode, but without the applied spark gap. In the 2013 version, we have developed a new way to compute the offset in such a case. The spark gap, in this situation, will be computed on a solid shape that will be used to create your "real" electrode shape.

"3 in 1": Creation of 3 electrodes in 1 step

Until now, TopSolid'Electrode allowed two ways to create your electrode:

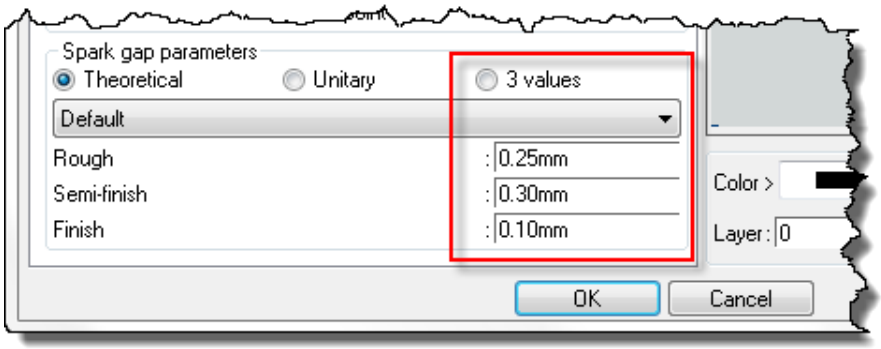
- **Use for the file**: It was the mode you used to create a "theoretical" electrode. The software managed three spark gap values used when dimensioning the electrode points of control. They were also useful in TopSolid'Cam and/or TopSolid'Draft.
- **Apply to the part**: It was the mode you used to create the "real" electrode, with one spark gap value applied.

When you needed to create the three "real" electrode shapes, with three different spark gaps values, you had to start three times the electrode creation function.

It is now still possible to manage the two previous modes, but they have been renamed for more clarity:

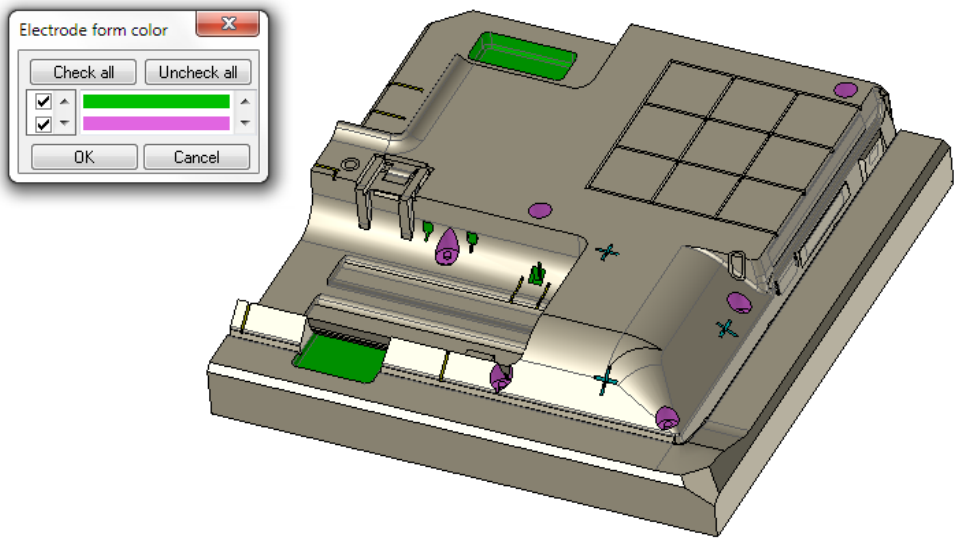
- **Theoretical;**
- **Unitary.**

A third mode has also been added: **3 values**. This mode allows, as a step by step wizard, to create the three electrodes, with three spark gap values, in a single operation:



Electrode creation on colorized faces


It is now possible to create electrodes on shapes which include colorized faces. When starting the function, TopSolid'Electrode analyzes the different colored areas (they can come from an import process or they can be colorized in TopSolid'Design using the function **Shape | Mechanical/Other operations | Color**) and allows you to create an electrode per area.



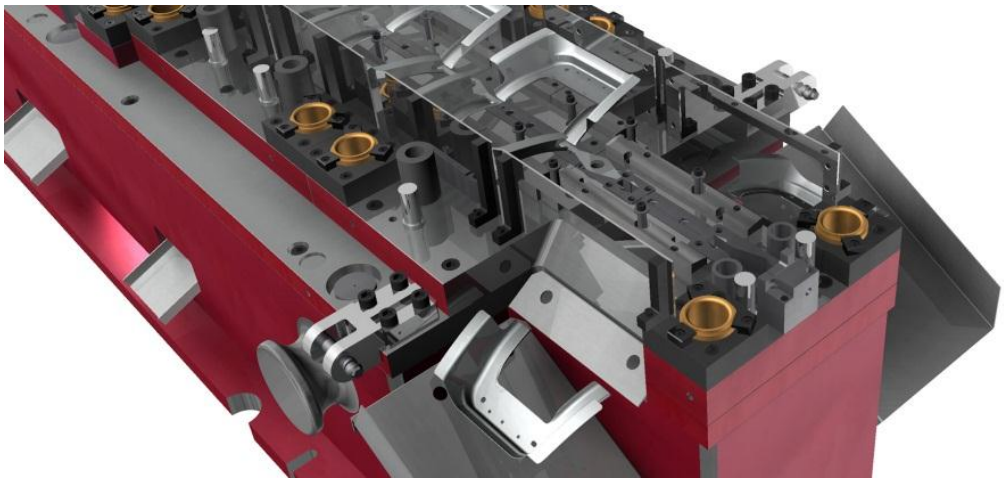
TopSolid'Electrode creates each electrode, as a step by step wizard. Of course, you can manage the parameters for each shape, according to the dimensions of the electrode shell.

Modification

Electrode shell update

It is now possible, by right-clicking on the shell in the design tree, to manage the update of this element. It allows you to add/remove faces from the initial shell, for example if the reference part has been updated, and if TopSolid'Electrode were not able to automatically update this shell.

TopSolid'Progress 2013: What's New



This document describes the improvements made to the **TopSolid'Progress** software: **2013** version.

General

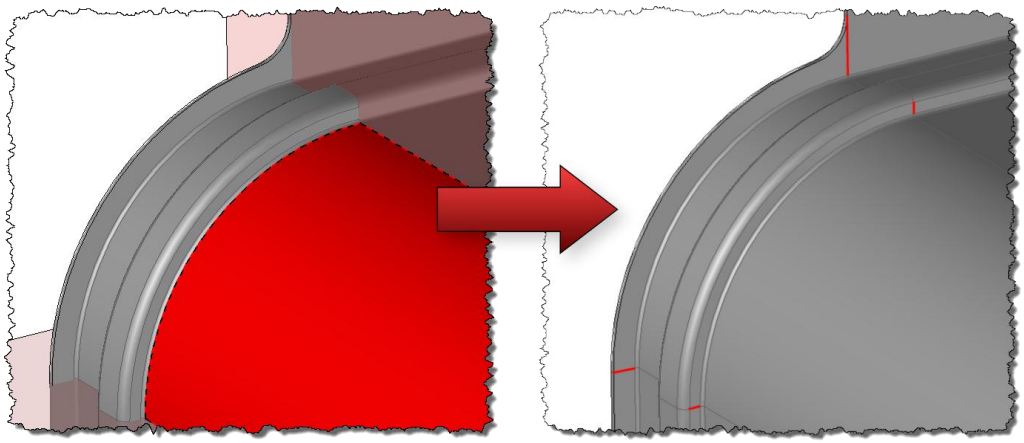
Tool set checking

 When a Progress document is loaded, a warning is now reported when some of the tool sets are missing ("punch set", "stripper set", "die set"…).
Most of the time, it happens when loading a TopSolid'Design document (*.top) as a Progress document (*.pgs).
If this type of warning occurs, the tool sets must be rebuilt using the **Progress die | Tools | Rebuild tool sets** command.

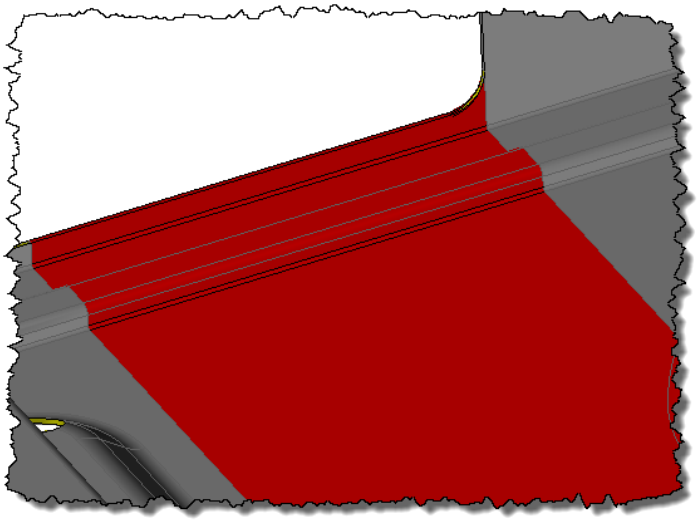
Part preparation

Isolate bend: Isolate bend with forming

 This new option is used to isolate the bends containing center or lateral formings, before using the **Unbend for intermediate stage** command.
It displays the preview of the planes that will be used to create the edges that will isolate the bend. The lateral extension length of the planes, as well as the extension radius of the imprint tools, can be adjusted with the **Overlength** and **Radius** parameters.

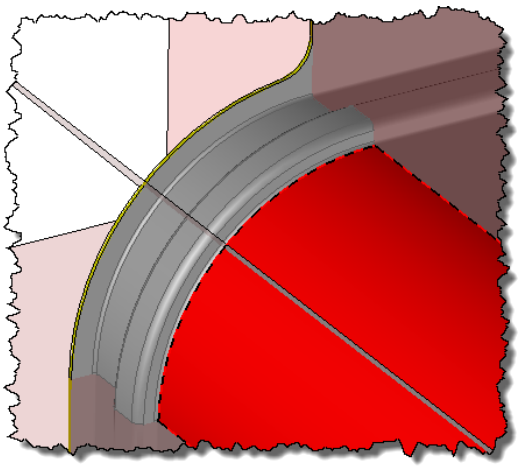


Preview of imprint tools and imprint results.



Result after an unbend for intermediate stage operation.

The **Divide** sub option allows you to divide the bend into several subparts in order to prepare a partial unbend.



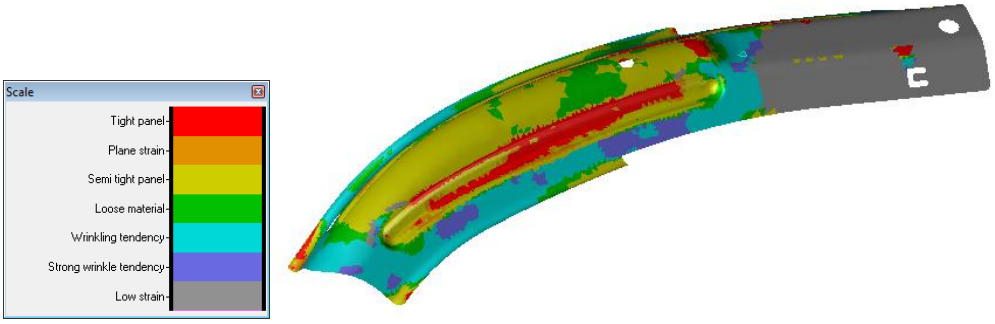
*The **Divide** option.*

Unstamping: Analyze unstamping results

The **Analyze unstamping results command** (FTI) now allows you to view:

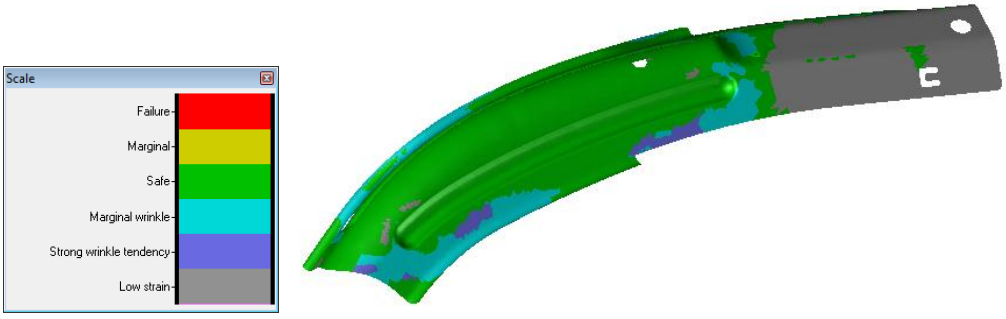
- The forming zones;
- The safety zones.

Displaying the **FORMING ZONES** enables you to view different areas on the part, according to the type of recorded strains depending on the combination and intensities of stretching and compression analyzed at each point.



Forming zone analysis sample

Displaying the **SAFETY ZONES** enables you to associate the analysis of the formings to the elastic limit criterions in order to view areas according to explicit feasibility criterions.



Safety zone analysis sample

However, this analysis has some limitations. It uses a constant thickness value for the whole part. So, it modelizes quite approximately the forming processes of parts made with several stages. The limit conditions of the forming process are ignored (influence of stripper plate, of holding stiffeners…).

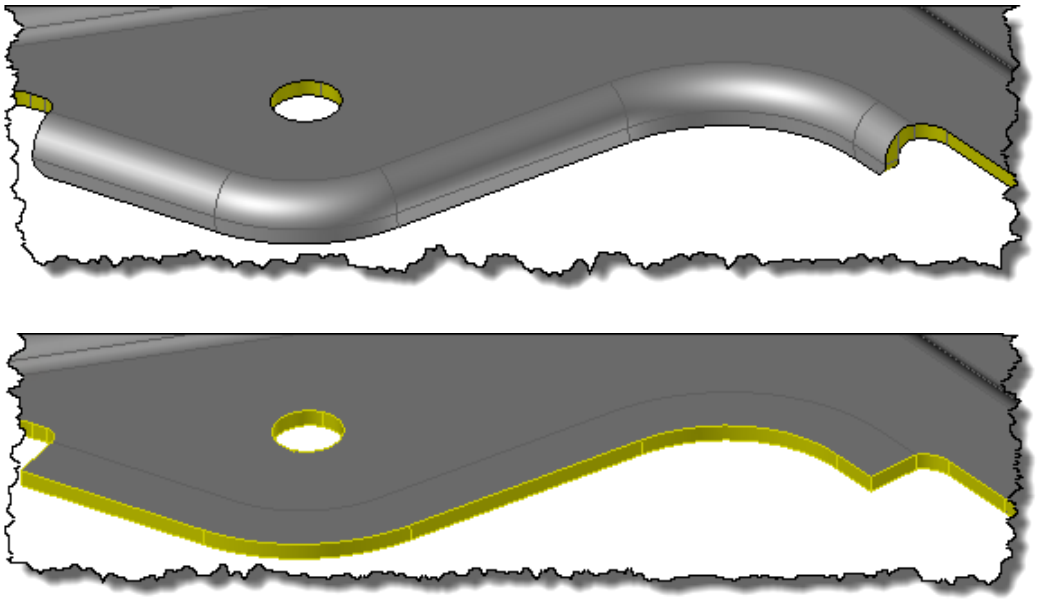
The results are only available for **unstamping simulation** made with the module of our **FTI** partner.

Flange unfold

 This new command is dedicated to unstamping/straightening of formings standing on the border of parts. It replaces user part geometries of an area by its unfolded geometry.

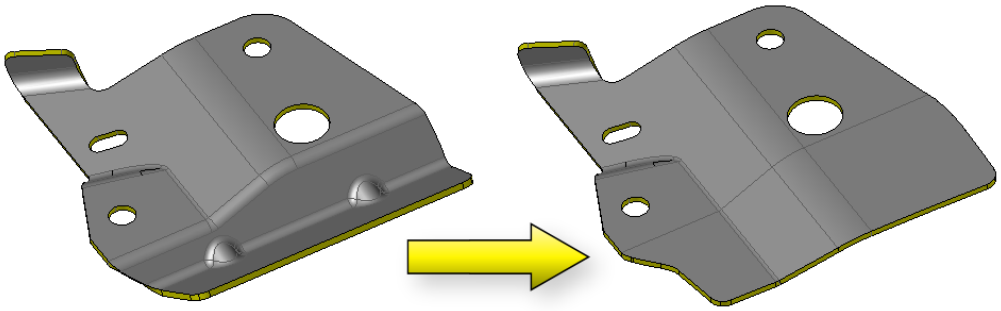
Two different modes are available:

- **EXTENSION**: this mode extends the replaced area by a linear extension of a given length.
  The holes standing inside the replaced area or the geometries at the border of the part are lost. The user must recreate it if needed.
  The option is made for simple cases.



*Sample of result given with the **Extension** option*

- **UNSTAMPING**: this mode makes a "real" local unstamping of the area to replace. Unstamping is made on a leading surface that may be computed automatically by the function, or given by the user (then, it must be created before).



With automatic leading surface mode, the surface that will be used is previewed before the execution of the operation.
The automatic leading surface mode is efficient when the boundaries of the area to replace are simple, continuous and without too big concavities (the created surface must have no self-intersections).

It automates tasks that were previously done with the following commands:

- **Cut part**;
- **Composite surface**;
- **Leading surface**;
- **Unstamping**;
- **Trim and thicken**.

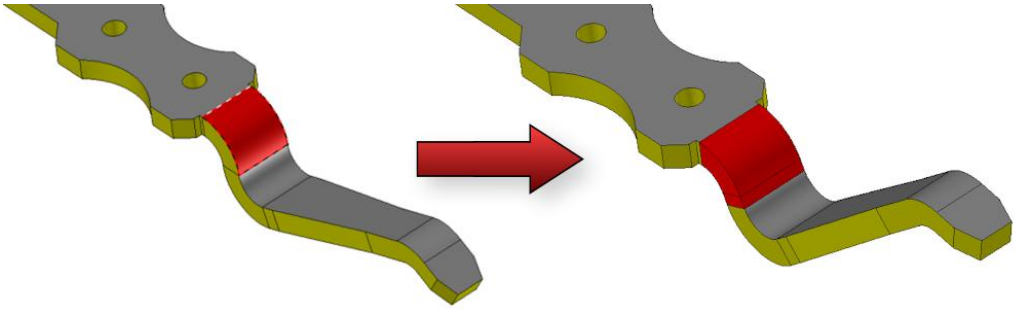
Of course, these commands are still available to treat difficult configurations.

This option is only available if you have the **Forming Simulation FTI** optional module.

Modify bend: Advanced management of overbending

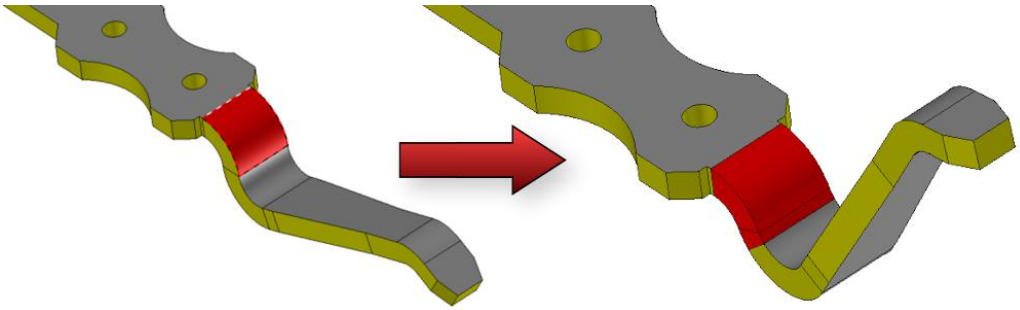
 In this command, several new options have been added.

- In the **MODIFY FACE ONLY** mode, it is now possible to compute the new bending angle according to the modified radius value, or to compute the new bending radius according to the modified angle value. These computation modes are similar to the computed modes of the **Overbending on profile** command.
- In the **ROLL ON NEIGHBOUR FACES** mode, the new option **Manage continuity** allows you to control how the additional geometries will be added, between the modified area and its connected areas. It the option is set to **NO**, the additional geometries will be linear.



Reduction of the bend length with a linear additional area.

If the option is set to **YES**, additional geometries will be cylindrical if the neighbor area is, and it will take the same axis and radius.



Reduction of the bend length with a cylindrical additional area, as the neighbor face.

Body management

 This new command is dedicated to the management of parts for which we want to move some sub parts relatively to the others. This is typically the case of some connector parts that contain unbendable sub parts linked by some forming areas.

In **Invert strip** methodology, this command will be used to isolate the unbendable areas, to reposition it in the part, and then to unbend it step by step. The forming areas will be removed to be replaced with some transition shapes designed with the help of the connected sub parts.

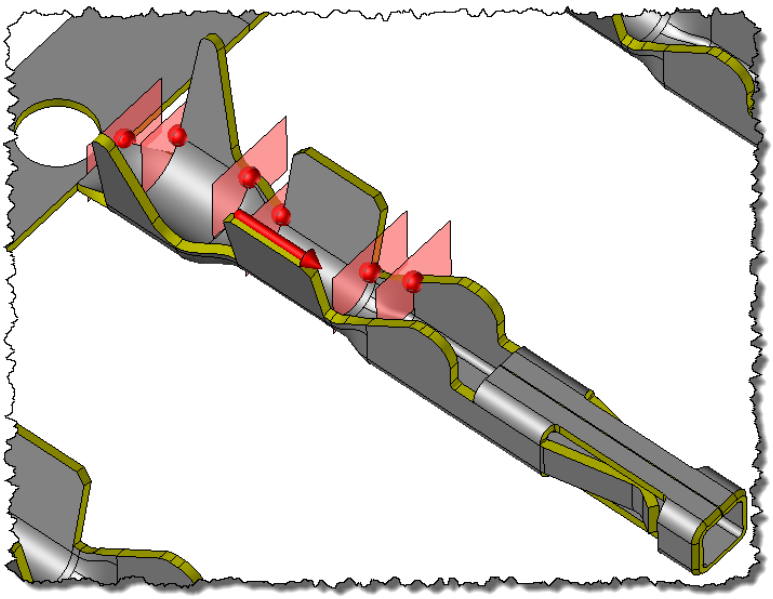
Five different subfunctions are available:

- **Divide**;
- **Move bodies**;
- **Remove bodies**;
- **Add body**;
- **Merge bodies**.

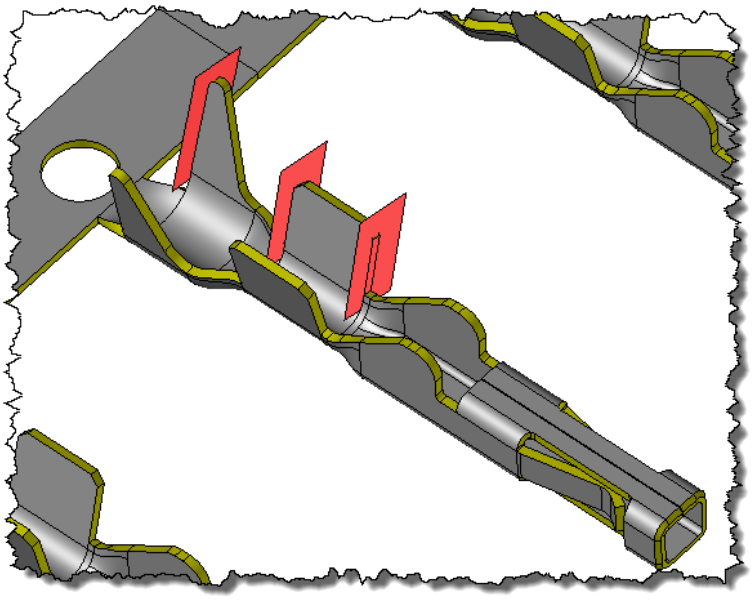
The **DIVIDE** function allows you to divide the part into several subparts.

Several division methods are provided:

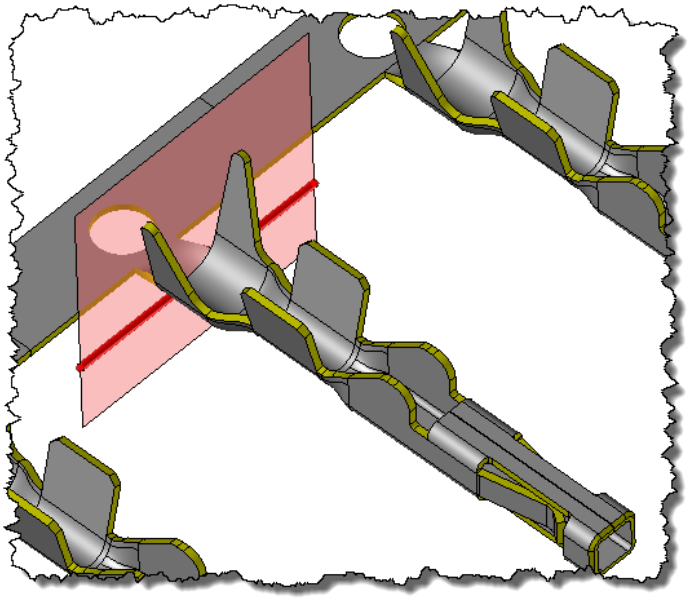
- By giving some points and a direction. The part will be divided along the planes perpendicular to the given direction and passing by the points.



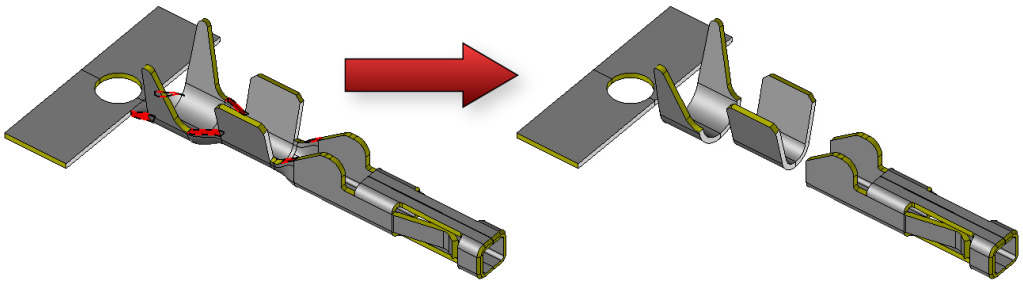
- By giving some planar faces parallel to each other. The part will be divided by the planes defined by the faces.



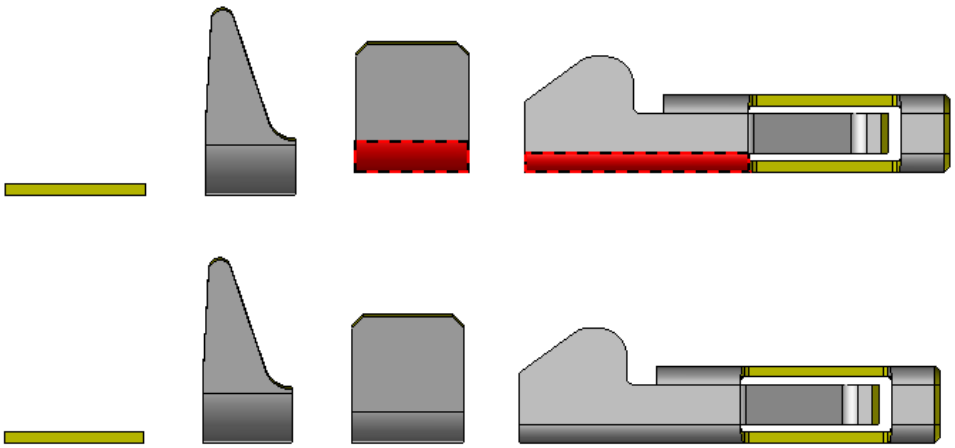
- By giving a trimming profile.



The **REMOVE BODIES** function allows you to remove some of the bodies previously isolated, most of the time, in order to replace it by their equivalent unfolded shapes.
The bodies are selected by one of their faces. It is possible to remove several bodies in a raw.

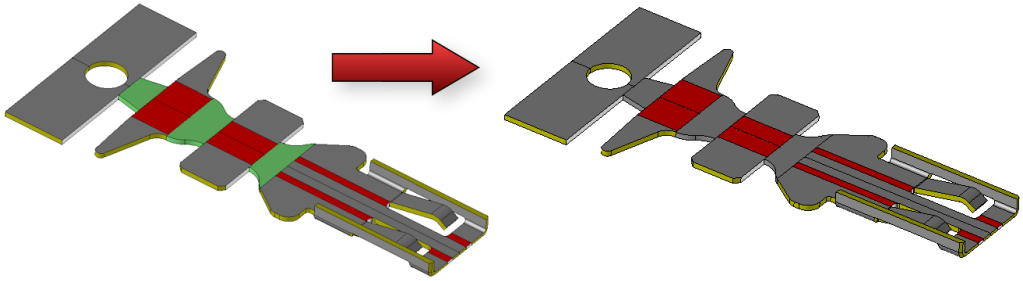


The **MOVE BODIES** function allows you to change the position of some bodies to apply the movement produced by the unfolding of the whole part.



Example of body displacement that reaches the unfold position.

The **ADD BODY** function is used to add new bodies to the part.
In general, these bodies are some flat transition parts, designed by the user with the help of subparts of the shape.

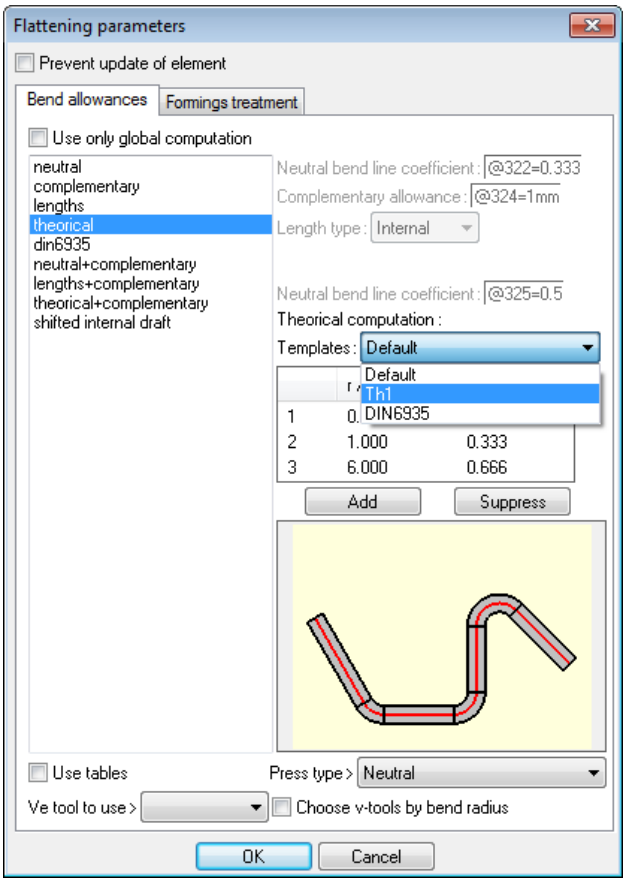


Finally, the **MERGE BODY** function allows you to unite the bodies together before importing the flatten part in order to create the final strip.

Strip

Import part: Templates of theoretical unbend allowances


In the **Import part** command, in the unfolding parameters, when selecting the theoretical bend allowances computation method, it is now possible to choose a template of method among several predefined templates (see **Tools | Options | Sheet Metal – Unbending |Compute bend allowance as**).

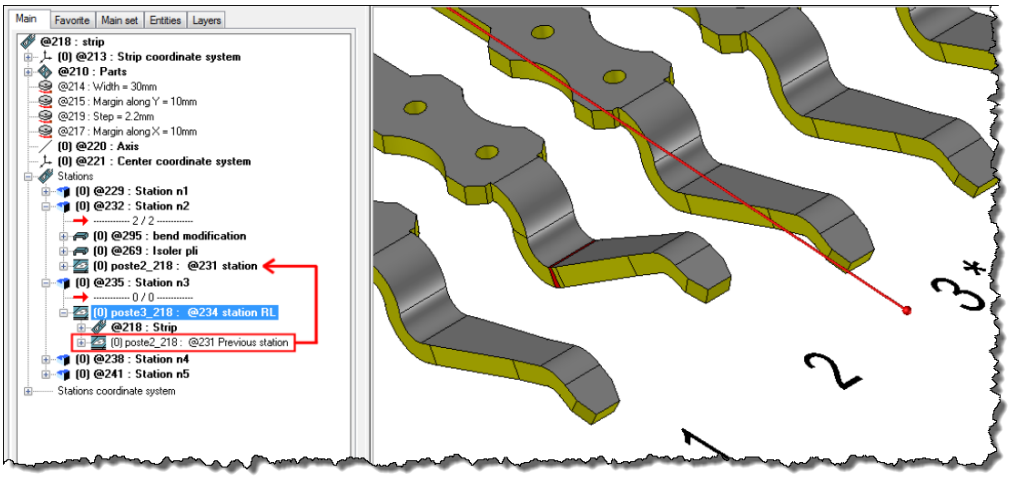


This option is also available in the **Sheet Metal | Generate flat pattern** command.

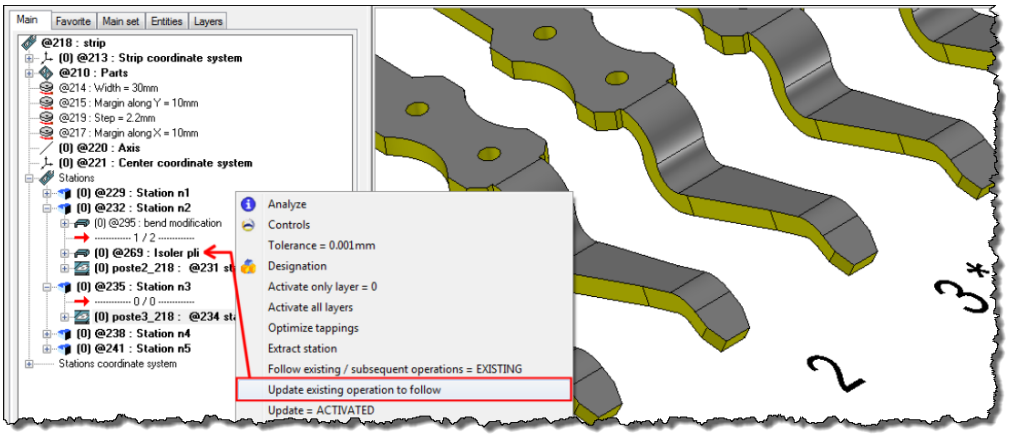
Strip edition: Follow existing/subsequent operations: Redefinition of the template operation

In the Operations tree, when you edited a strip, it was already possible to manage the **Follow existing/subsequent operations** option on the initial definition of a station.

When using the **Follow existing operations** mode, it is now possible to redefine the operation used as template on the previous station using the **Update existing operation to follow** contextual command.
For instance, in the screenshot below, the station 3 is in **Follow existing operations** mode, based on element @231 of station 2.



The **Update existing operation to follow** command will redirect the template of the followed operation on the last operation targeted by the insertion operation cursor.
In the screenshot below, the result of the isolate bend operation @269 will be then applied to the station 3.



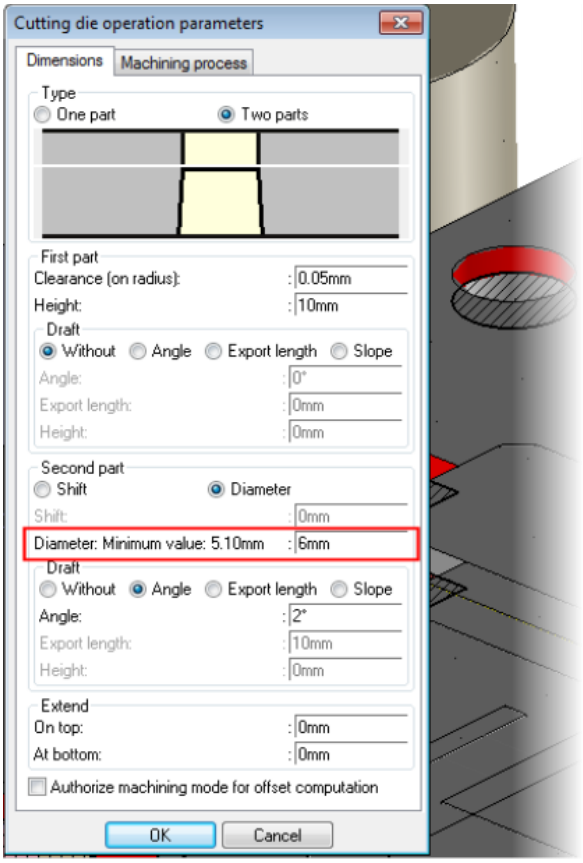
Free bending

In the **Free bending** command, it is now possible to apply the bending movement to another body of the part.
This configuration of use may occur when a strip is created with the **Multi bodies strip** option in a general process of invert strip design, and when wanting to change the orientation of a part in the strip (in general, when wanting to prepare a forming process).

Tools and dies

Cutting die: Dimensions of the second part by diameter


In the **Cutting die** command, when a circular profile is selected, it is possible to define the dimensions of the second part of the die with an explicit diameter value. It allows you to define rounded values, more easy to treat for machining.



Corner relief: Selection of edges to treat


In **Symmetrical wire**, **Fillet** and **Chamfer** modes, when the edges to operate are selected in **loop** mode, now it is possible to exclude some of the selected edges.

Tools

Duplicate for machining


The export format of the generated file can now be configured: ***.pgs** or ***.top**.
The new available format ***.top** shall be preferred to send a part for machining on a TopSolid'Cam station that does not have TopSolid'Progress, or to import the generated file in TopSolid 7 (that only read V6 file with the extension ***.top**).

TopSolid'Wood 2013: What's New

This document describes the improvements made to the **TopSolid'Wood** software: **2013** version.

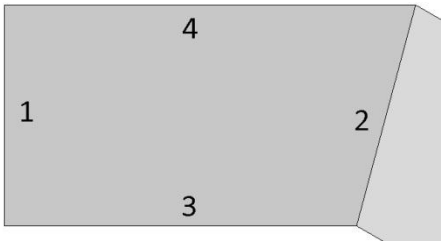
Constrained block version 2

The new **Allow non-parallel face** option is now available in the **Constrained Block** function to help you create the part between non-parallel planes.

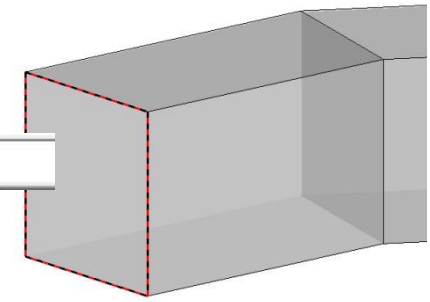
This option makes possible the creation of triangle and trapezoidal parts.

Example: Non-rectangle part

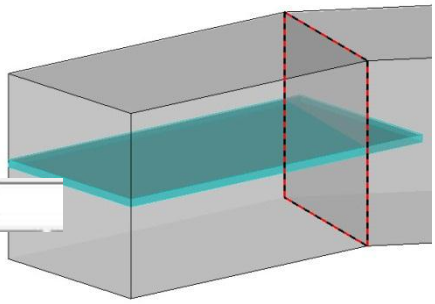
Warning: If you have 2 parallel planes and 2 non-parallel planes, you need to start by the non-parallel planes.



- Enter the first shift and select the first non-parallel face.

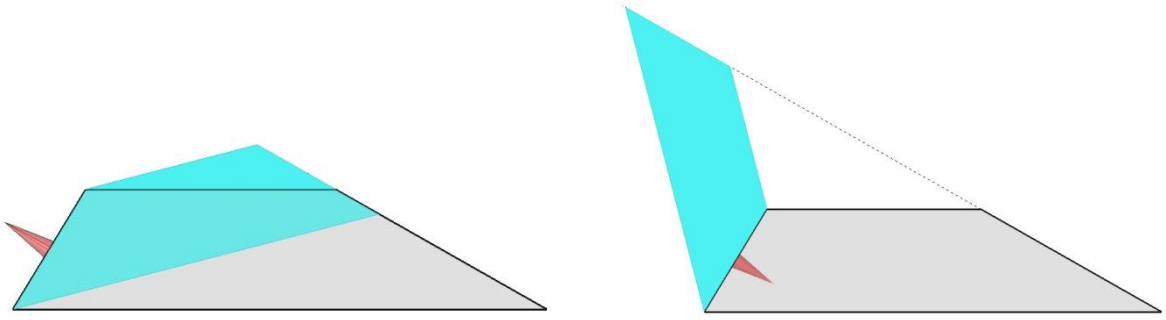


- Switch to non-parallel mode: **Allow non-parallel faces = YES.**
- Enter the second shift and the second non-parallel face.

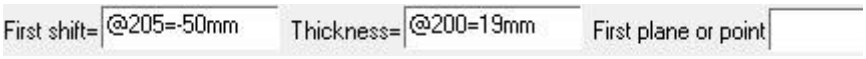


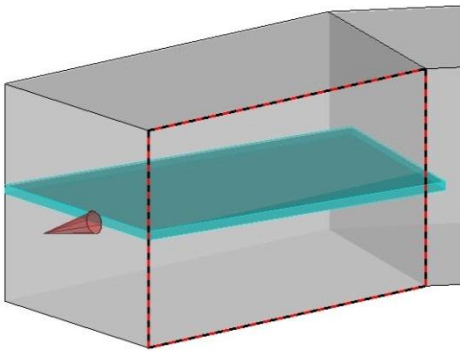
Note: It is not possible to use the Length mode in non-parallel face option to get the second face position.

A red arrow will appear: it allows you to change the direction if the constrained block refers to the first face selected.

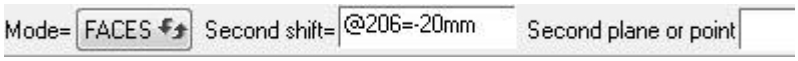


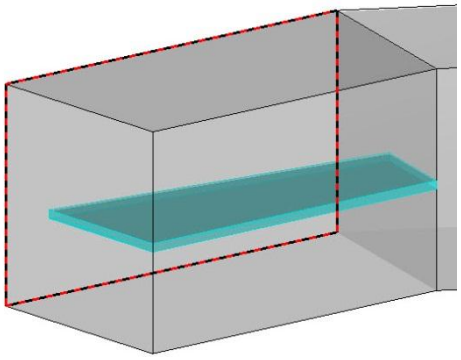
- Enter the third shift and select the third face.





- Enter the fourth shift and select the fourth face.



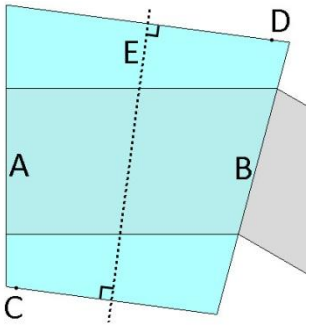


Here it is possible to change to Length mode and enter the distance value.

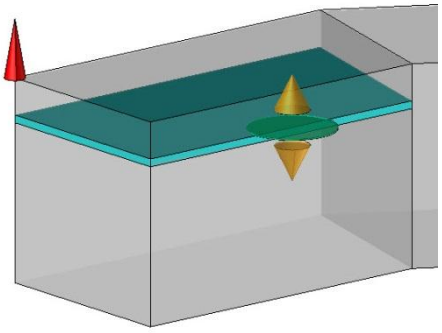
In this mode, the fourth face should be parallel to the third face selected.

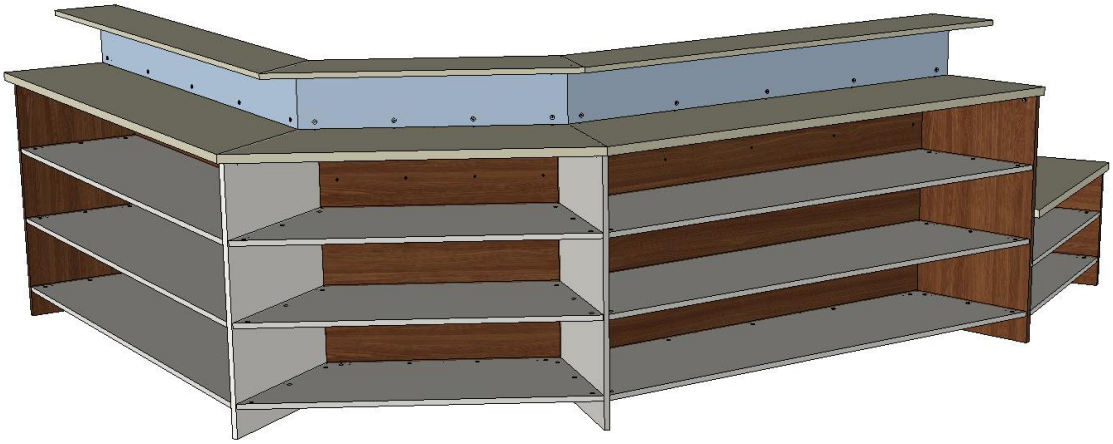
After entering the length, the red arrow will appear; you can click on it to invert the direction of this constrained block.

It is possible to select the points C and D to create the third and the fourth faces. The third and fourth faces created will be perpendicular to the bisectrix (E); this bisectrix is created between the faces A and B.



- Finally, you can place this part using the distribute mode or linear constraint, or just select one face and enter the offset distance.

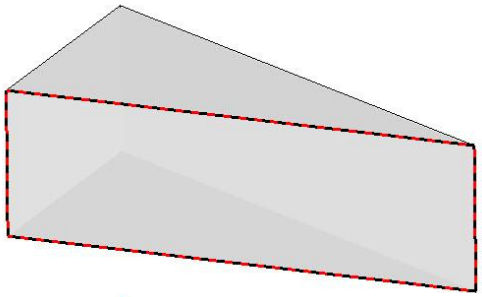




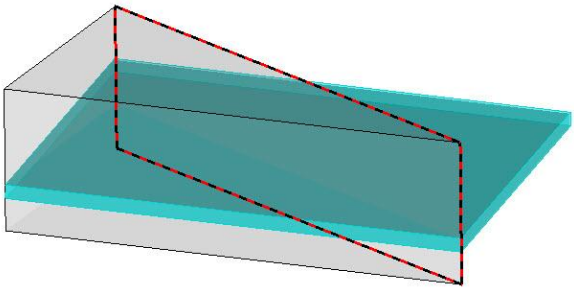
Example: Triangle part

The configuration principle is the same as the << Non rectangle >> example, except for the fourth face, you select a point.

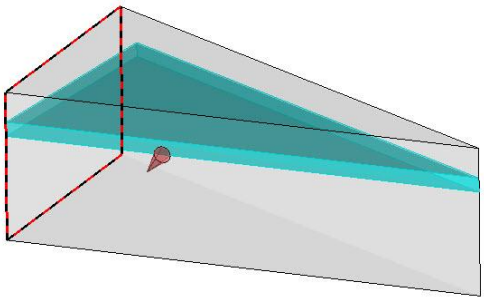
- Select the face of the triangle part.



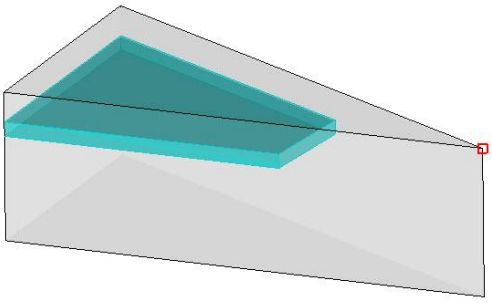
- Select the second face of the triangle using the **Allow non-parallel faces = YES** mode.



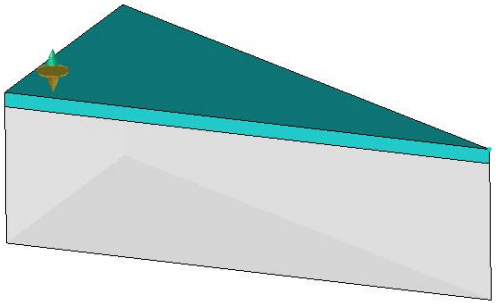
- Select the third face of the triangle.

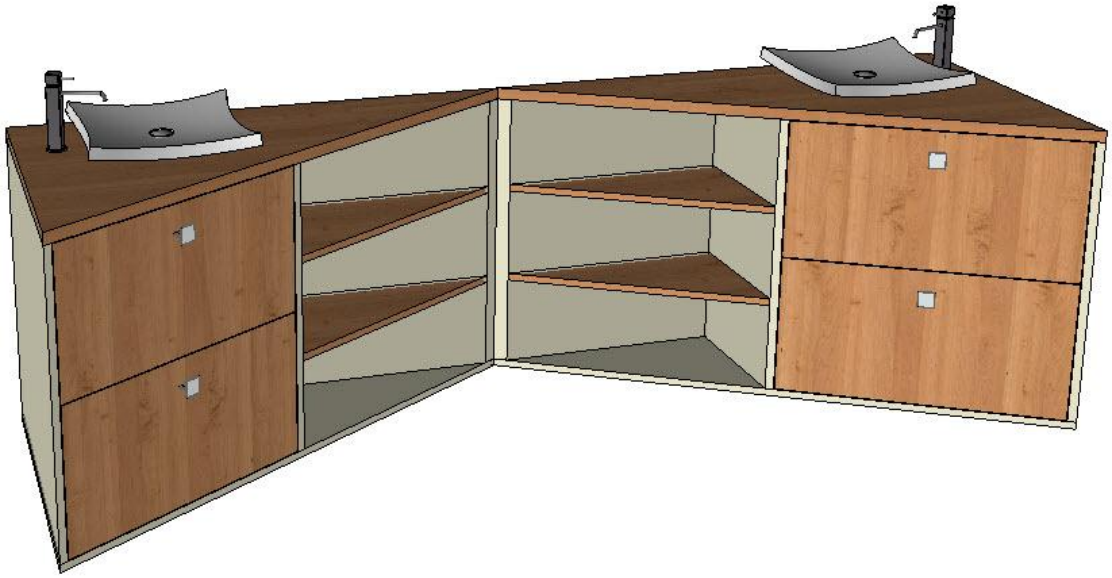


- Select the intersection point of the face 1 and face 2.



- Select the face to give the position.





Driver block version 2

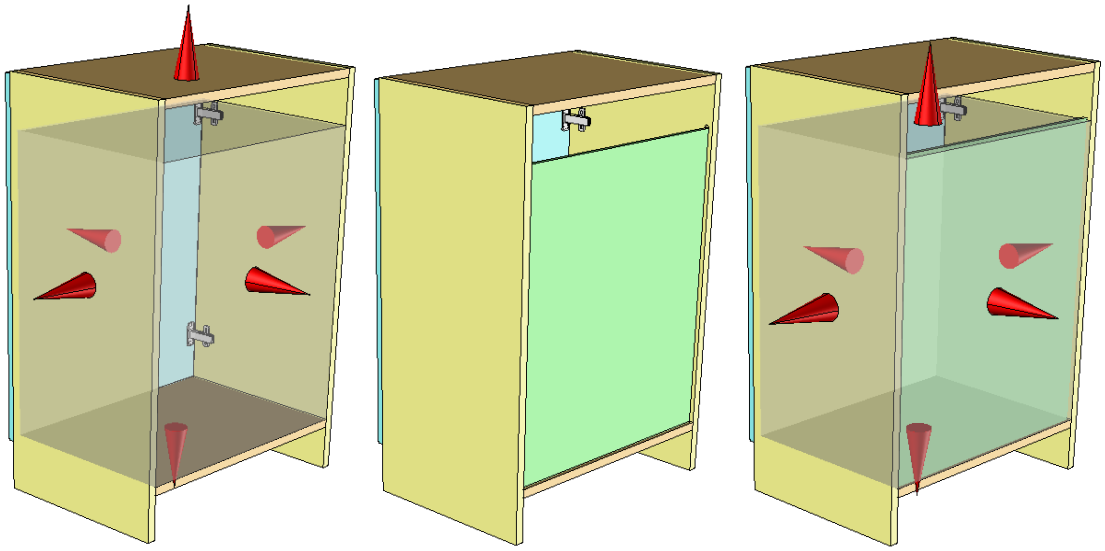
The **Driver Block** function has been enhanced with new operations in the 2013 version.

Adjust position

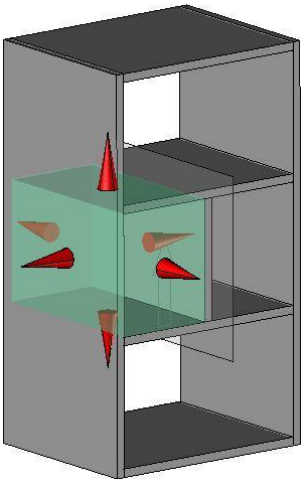
You can adjust one face of the driver block just by clicking the red arrow:

- You can input the shift distance to adjust the position: the faces can adjust from original face with the input value.
- You can input the length: the face you adjust will have a distance from the opposite face.

It is possible to input two different adjust distances on the opposite side, but you cannot input two lengths to control on part.



Reminder: Give preference to publishings



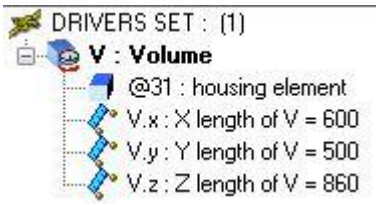
When inserting the component with driver block or when modifying it, it will always take the publishing face at first.

When inputting the component with driver block if you have the publication on the face, the driver Block will put automatically on the publication, if not, it will position the component with the first face fund.

Warning: In order to avoid cyclic references, you should create the automatic Publishing on an alternative set and not on main set.

Modifying the name and the designation of the Driver Block

Now we can give and modify the Driver Block with a different name and the designation. It allows having a name to explain the utilization of the parameter and the have a designation refer to the position method.



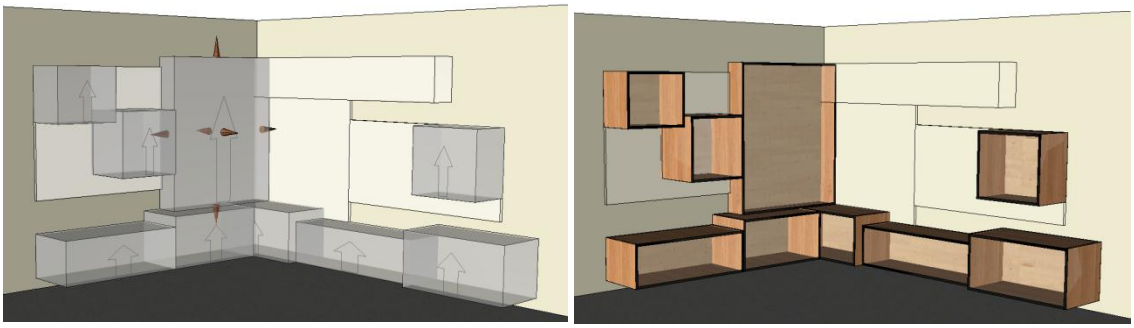
Multiple inclusion

Now it is possible to select several houses to insert the same Driver Block component just in one time. The other additional parameter inputted will be the same for all the insert components.

- To use this operation, insert your component, and select the first houses of Driver Block
  If you want to insert several Driver Blocks, select **Yes** to **Multiple inclusion**.

- Then, you can change the Housing mode for the next Driver Block in the rolling list **Housing Mode**, and then click on the arrow of the previous driver block to change the position face or give them the adjust distance and Length, or select the Box to position the next Driver Block.

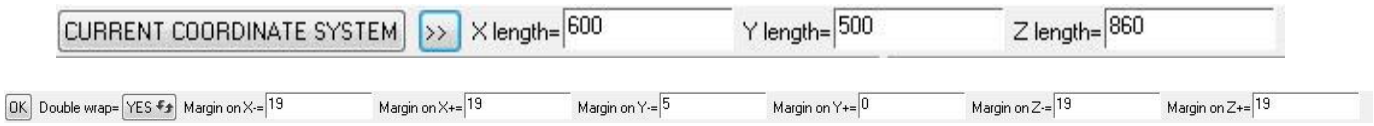
- Once all the Driver Block are inserted, validate with **OK**, then you can input the common parameters for all the inserted Driver Blocks.



Double wrap

The double wrap is now available in the Driver Block component to manage the adjust distance of the thickness of the part to the destination Cabinet.

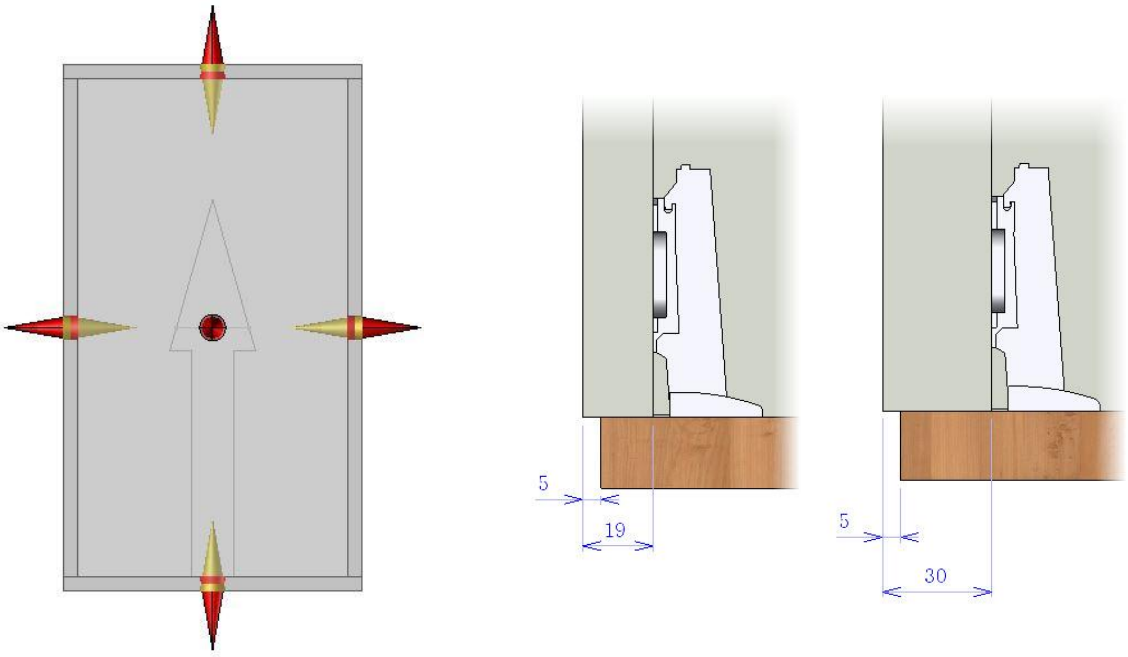
This type of the new Driver Block can be created by clicking on the double arrow **>>**: when you create the driver block



Just like the Driver Block component, the input value for margin is just used to create the model; these margin values will be recalculated with the thickness of the part of the destination Cabinet where we insert the component.

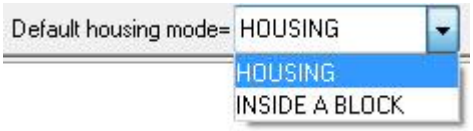
Name	Nominal value	Designation
Housing.x	600mm	X length of Housing
Housing.y	500mm	Y length of Housing
Housing.z	860mm	Z length of Housing
Housing.x1	22mm	X- thickness of Housing
Housing.x2	22mm	X+ thickness of Housing
Housing.y1	0mm	Y- thickness of Housing
Housing.y2	0mm	Y+ thickness of Housing
Housing.z1	22mm	Z- thickness of Housing
Housing.z2	30mm	Z+ thickness of Housing

When inserting the Driver Block with the double wrap, the inside face of the double envelope can be repositioned by clicking on the yellow arrow inside. (Those red arrows are the outside face of the driver block).



Defining the default insert mode of the Driver Block

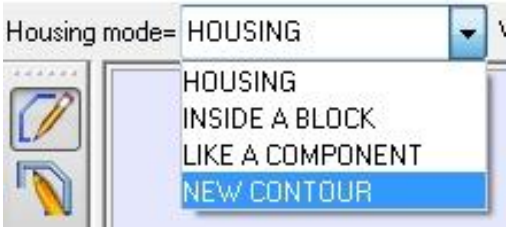
It is possible to define the default insert mode in the creation step of the Driver Block.



It is always possible to use another insertion mode than the default one when you insert your component.

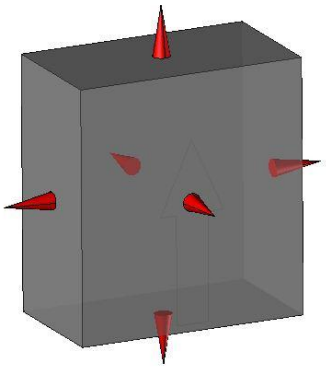
Creating a new Block when inserting the Driver Block

When inserting the Driver block component, one new option allows you to create directly a block, where you can insert your component, once you click on the option new contour, the function **Contour** is started.



It is possible to create a new contour. It is also possible to create the contour by passage Mode:

Once the contour is created, the function extrude will start automatically, you just need to input the value to give the height of block where you need to input the component.

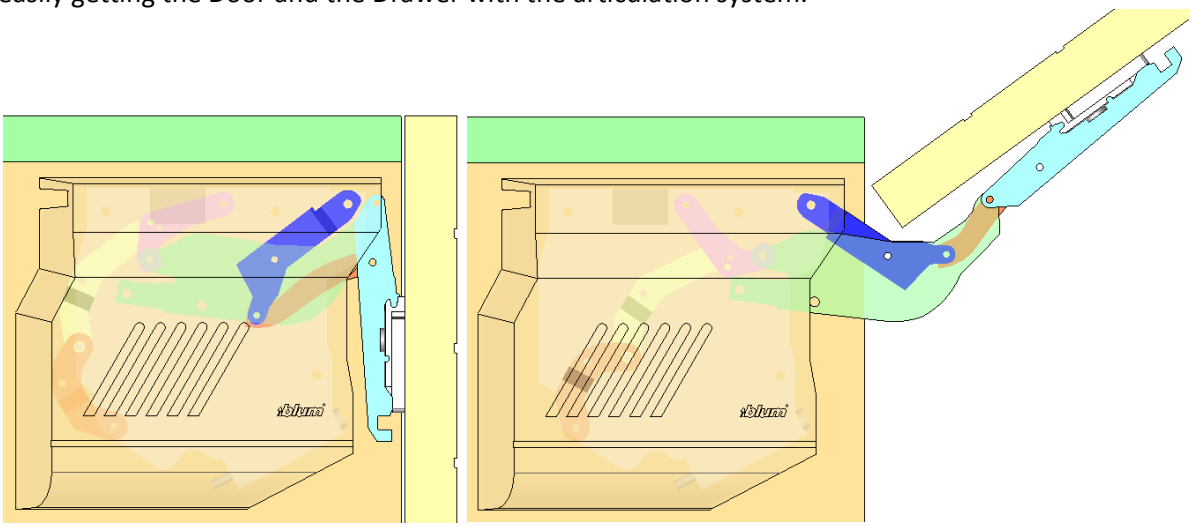


Hiding the insert box

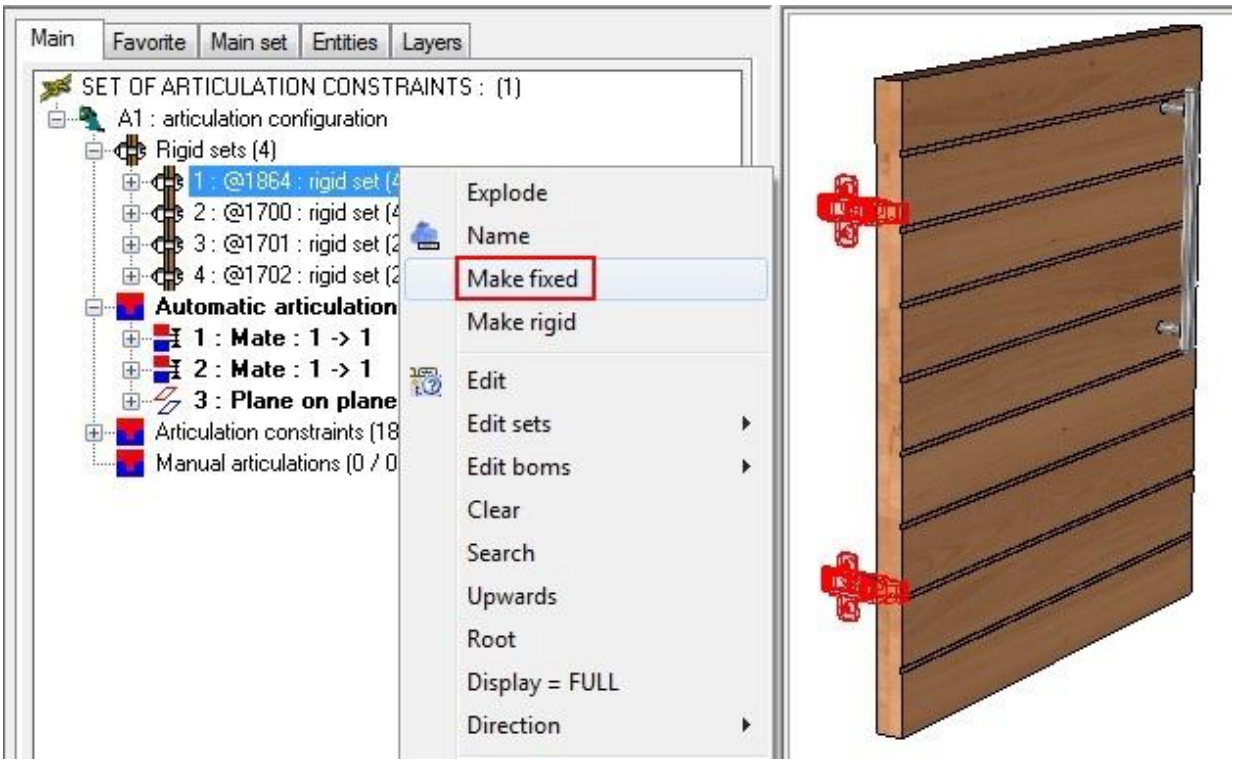
From today, you can directly hide the insert box just by modifying the **Hide block** mode to **Yes**.

Driver block and articulation

The new Driver block of the 2013 version is now compatible with articulation configuration. This improvement allows easily getting the Door and the Drawer with the articulation system.

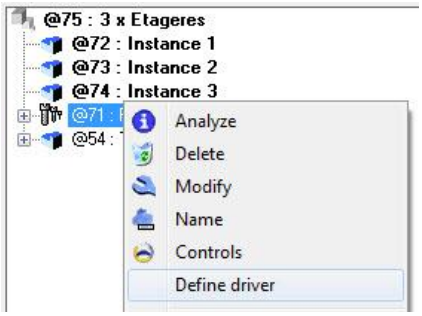


You just need to right-click on the element don't move to make it fixed because the driver block cannot add automatically the constraint. For example as this door, you should right-click on the Hinge pivot and **make it fixed**.



Propagation driver

Now it is possible to define the propagation as a driver.
This option allows you to modify locally an instance of the repetition, for example in order to avoid the collision of separation and hinge pivot of doors.
This new option is available for linear and circular and Wood propagation.

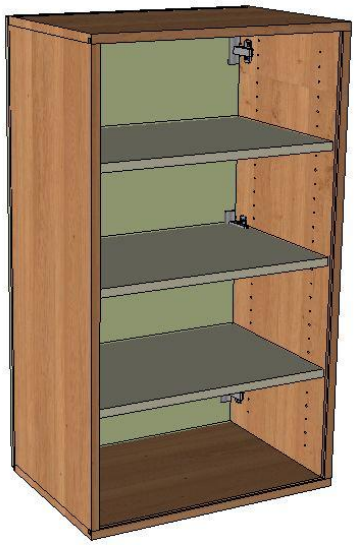


Creating the propagation driver in the component

- Edit the repetition in the construction tree, then in the repetition's propagation, you can define it as driver using a right-click.
- Define the propagation driver name.

- Define the propagation driver designation

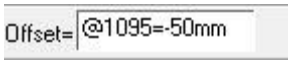
Using the propagation driver in the assembly



Example: A cabinet has a collision between the Hinge pivot and the separation.

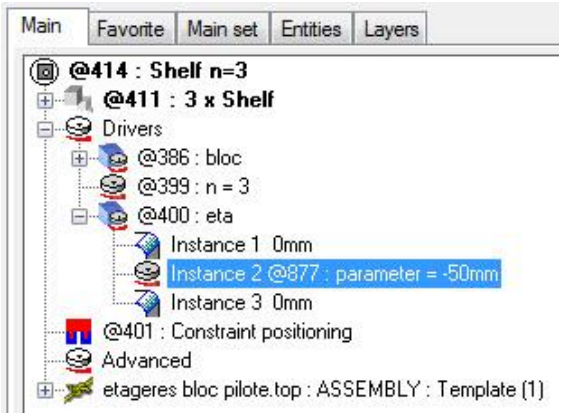
- Modify the instance of the separation (**Modify** function, and then click on the separation).
- In the dialog bar, select **OFFSET instance.**

- Then give the offset value.



Note: The positive value makes the offset in the propagation direction and the negative value makes the offset in the opposite direction of the propagation.

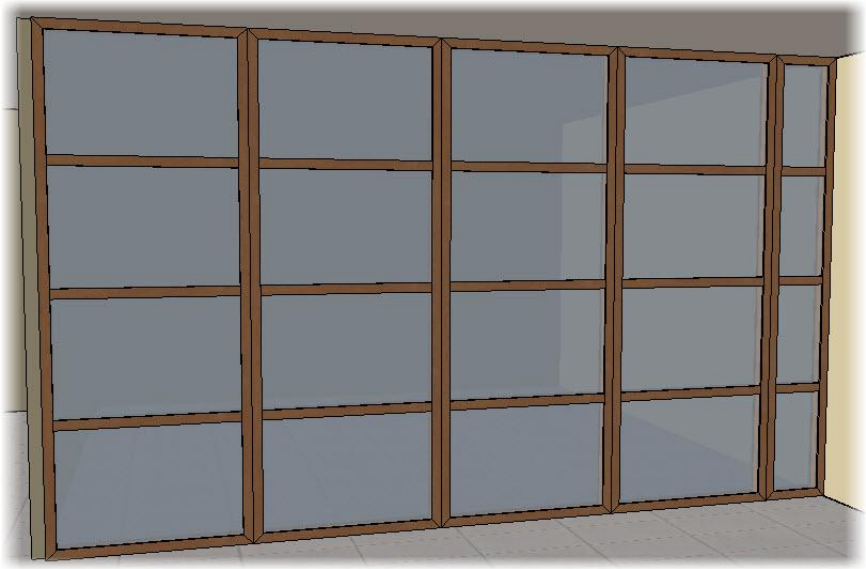
Of course, you can modify this propagation from the construction tree.



Distribution of the component

The new function of the distribution allows distributing the component between 2 elements.
In this example, the separation between 2 walls will change automatically the length of one of the instance to adapt the total length.

In this example, the separation is inserted with the width 1000mm and the dimension of the last instance is changed automatically to adapt the total length.

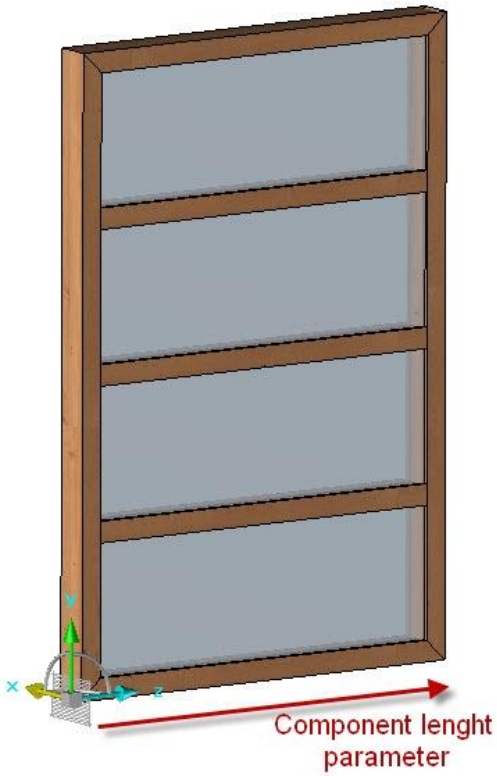


To define this type of component, you need to use the **Assembly | Define component | Define distribution position** command.

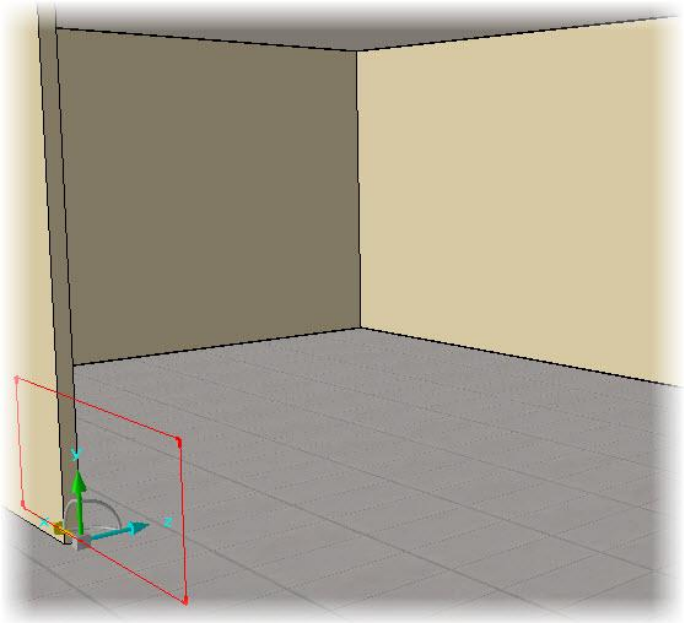
The component distribution will be defined by two elements:

- One positioning coordinate system: The coordinate system must be linked to the variable dimension of the component. And the direction of the Z+ must be the variable direction of the component.
- You must have one parameter to change the dimension.

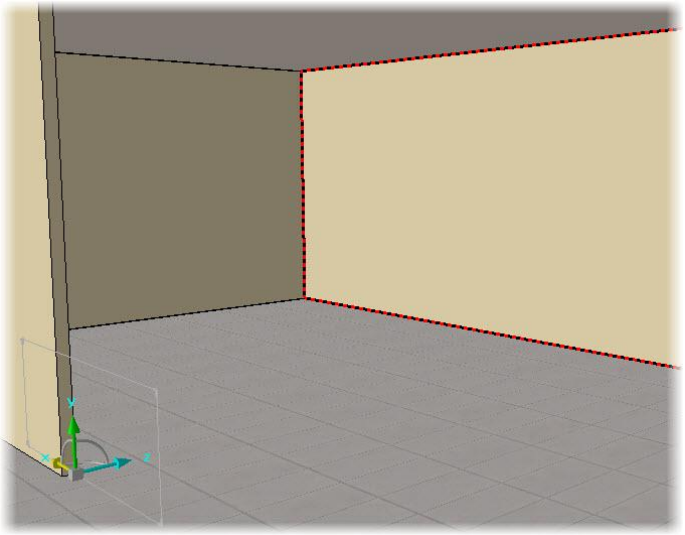
Then, when you include the component, the new **DISTRIBUTION** option allows you to adjust the dimension of the component between 2 planes.



Once you have selected the **DISTRIBUTION** option, you have to select the coordinate system to start it. The selected coordinate system must have the same direction as the component. It means the direction Z+ should be in the direction Z+.

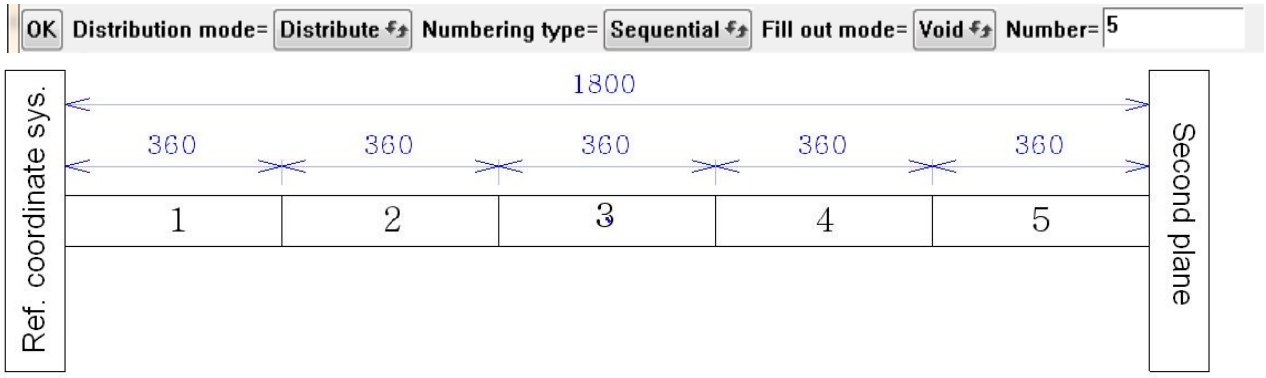


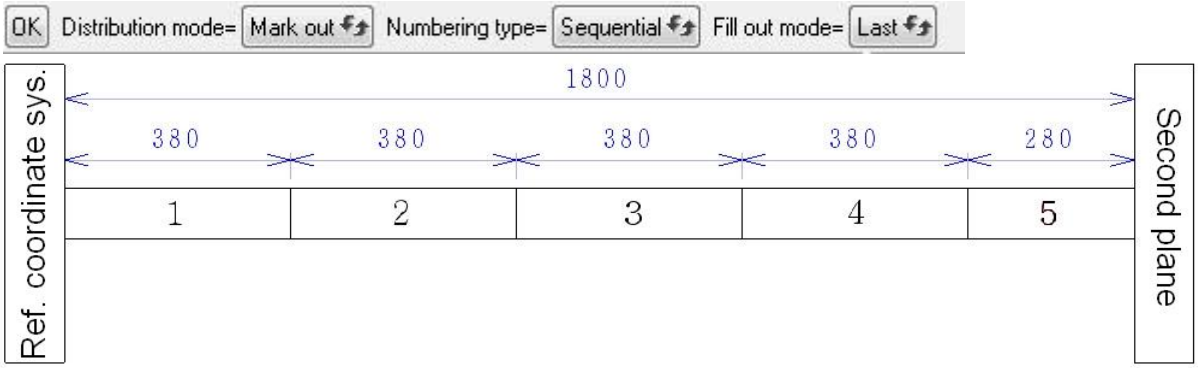
Then, you need to select the arrived coordinate system; it is possible to select a coordinate system, a face or a point.

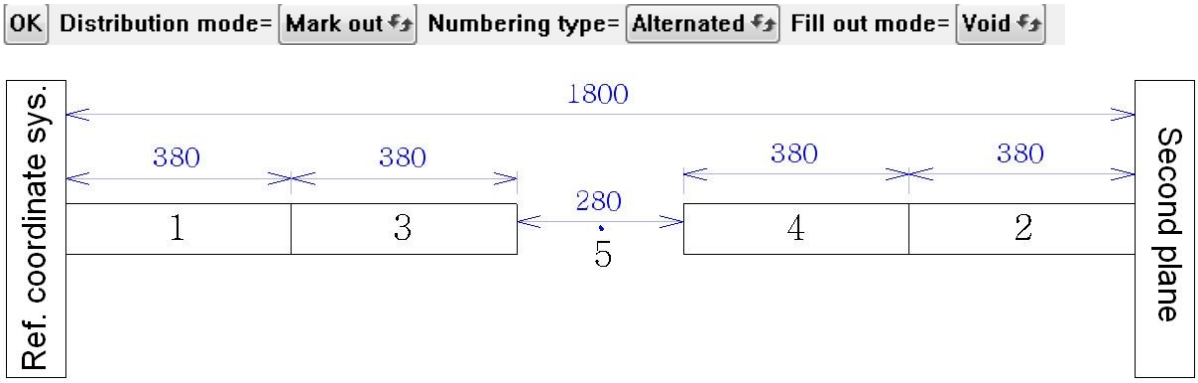


Once the second plan is selected, the distribution parameters will appear:

- **Distribution Mode**: **Mark out/Distribute.** In the mode **Mark out**, only the last instance of the component will change the dimension to adapt the distance. In the mode **Distribute**, the entire component will change the dimension to fill the total distance.
- **Numbering type: Sequential/ Alternative.** In the Mode **Sequential,** The modified dimension will be positioned at the end of the repetition. The Mode **Alternative** will modify the middle component.
- **Fill out Mode: Last/Void.** Allows deciding if the component modified is included (mode **Last**) or not (mode **Voided**).



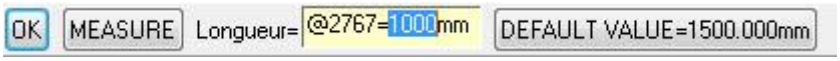




When you modify the distribution:

- **OK:** Allows you to validate the modification.
- **Component:** Makes changes to the selected component. For example, you can interchange the component by another one.

Interchange: Once you have selected the **Interchange** function, you need to select the component to be inserted. After inserting the component, it is possible to modify the length value. By clicking **OK,** the length will be calculated by the distribution in the assembly. Using the **Measure** function, you can measure the distance in the assembly, or you can use the **Default value** button. The interchanged component will take the length value of the component.



Note: To interchange the distribution by another, the reference coordinate system and the parameter used

for distribution does not need to be defined in the same way. You just need to have the distribution in your assembly.

- **Distribution parameter**: Allows you to modify the parameter of the distribution.
- **Reset**: Resets the default value of the distribution.
- **Component:** Allows you to modify the component model. In the advanced options (**>>**), you can change the model of the entire component or only the selected component, and the component not yet changed.

Extruded component on profile

When inserting the extruded component, this new option allows you to directly create the profile, and especially the profile made up of several lines.

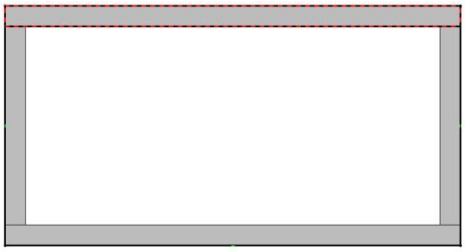
When inserting the extruded component from the **Wood | Other processes | Extruded component** command, this option is available using the **On curve** button after selecting the component to be placed.

- Select the contour (or the contour with the multi-selection) on which you want to position the profile. It is possible to create the new contour with the **Profile** or **Sketch** modes by clicking the **On curves** button.

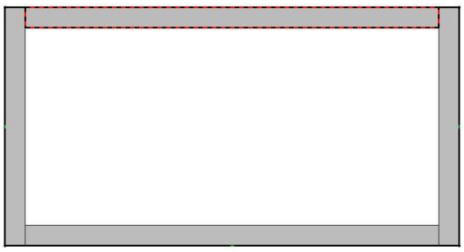
- Once you have selected the curve/sketch, set the profiles options :

  - **Key Point**: Allows you to choose the key point of the profile, where you want to position the profile.
  - **Rotation angle:** Allows you to define the rotation angle for the profile.
  - **Cutext:** Allows you to select the automatic cut mode for the entire component.

  - **None:** No cut will be applied to the extruded component.
  - **Miter cut:** MITRE cut for all the profile.
  - **Cutext key covering/covered:** Each component will have one side covered and one side no covered.
  - **Cutext Key first extruded covering/covered:** The first profile component will have 2 covered or 2 covering components.

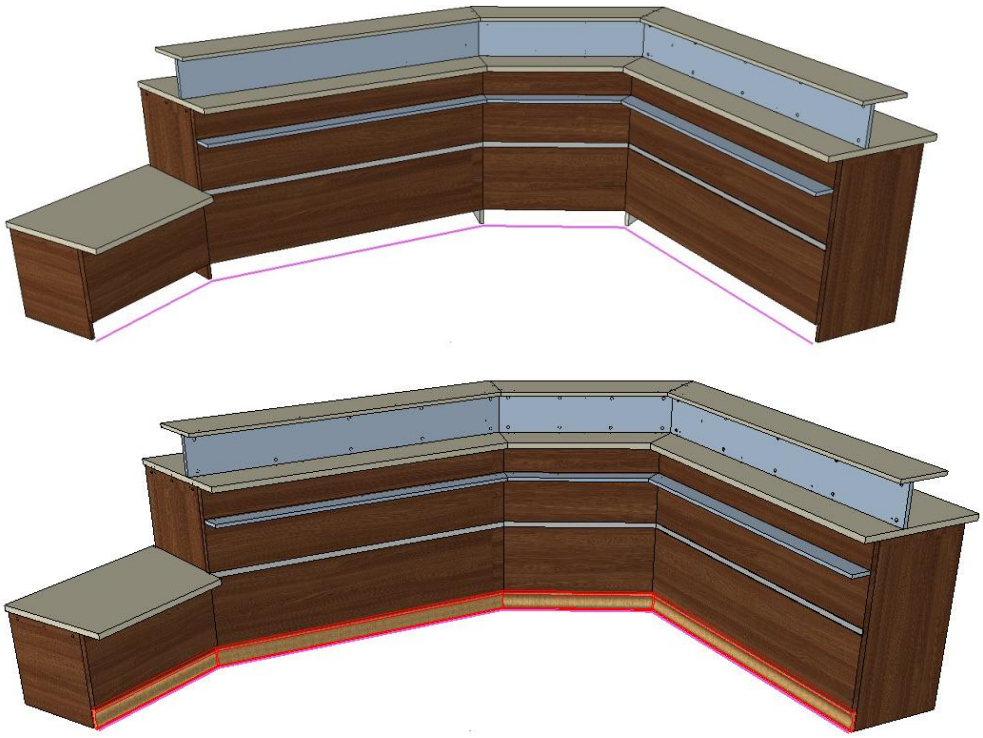


First profile covering



First profile covered

- Once all the parameters are set, validate by clicking the **STOP** button to include the profile.



Here all the plinths are inserted just on the new profile.

When modifying the profile :

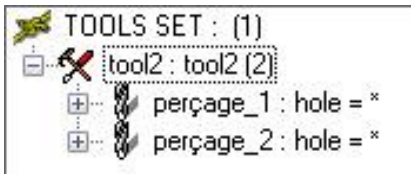
- **OK**: Allows you to validate the modifications.
- **Component**: Allows you to modify only the selected profile, and not all the profiles.
- **Reset**: Allows you to reset all the modifications to 0 (template, key point, rotation).
- **Template**: Allows you to change the template of the inserted component.
- **Add/Remove curves**: Allows you to add and remove the profiles in multiple mode.
- **Key Points/Rotation angle/Cut**: Allows you to modify the parameters on all the included profiles.
- **Advanced Options >>**: Allows you to choose whether the modifications of parameters, key points, rotation angles will be applied to the entire instance, or only on the instance which has not been changed yet (from the **Component** button).

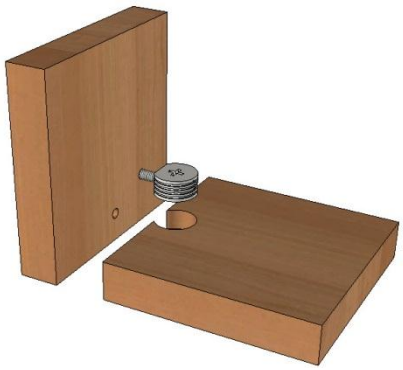
Restrictive rule of automatic processes

This new option is available when creating/modifying the tools of your component and allows you to choose which part you want to cut when activating the processes.

Before, when activating the automatic processes, all parts that collided with the tools were cut.

Example: The tool creates two drillings in this component.





If this component is inserted in the assembly where you have two parts joint them together. The drilling of the case find the collision of two parts, it will automatically cut two parts will the drilling processes:

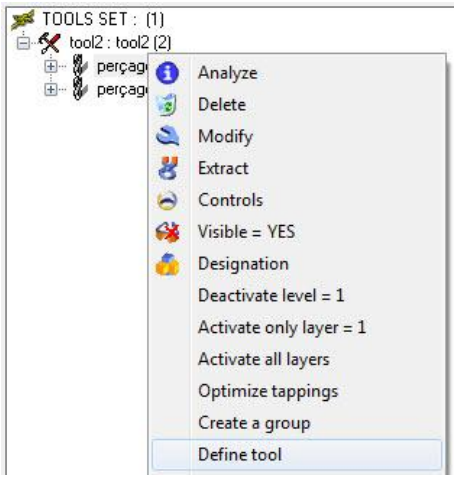
From now, in the assembly template, if a volume (Cylinder or extruded shape) is created in the place where we want create the drilling of the case, only the part that has the collision with the shape will be cut.

Creating the extruded shape:

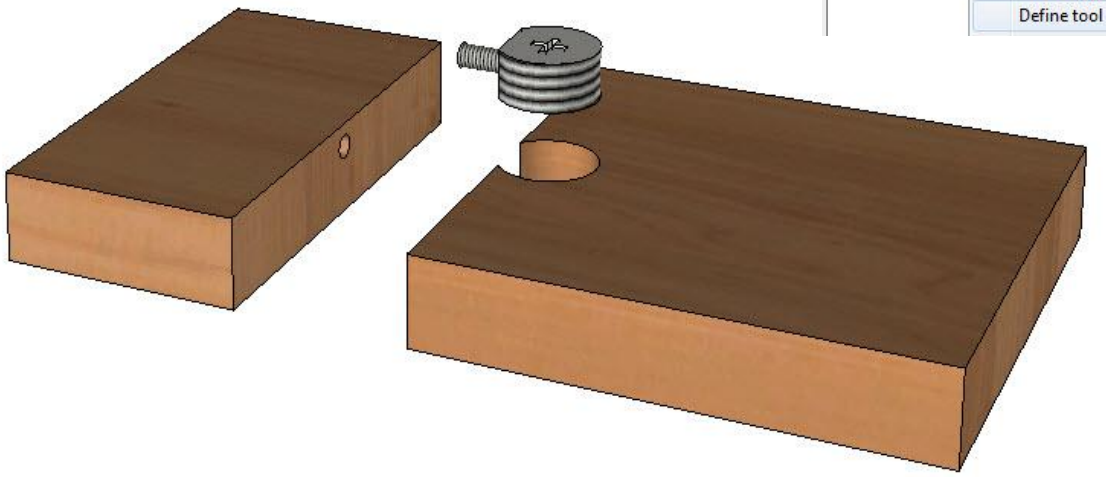
The shape (shown in green) limits the drilling of the case on one part only.

Link the limitation of drilling with the shape

- In the construction tree, **right-click | Edit set**, and then **right-click on the tools | Define tool**.
- In the dialogue bar, select **Define Clash shape**, and then select the extruded shape.



The extruded shape is link to the drilling operation now. The drilling operation is only works on the part that have the collision with the extruded shape.

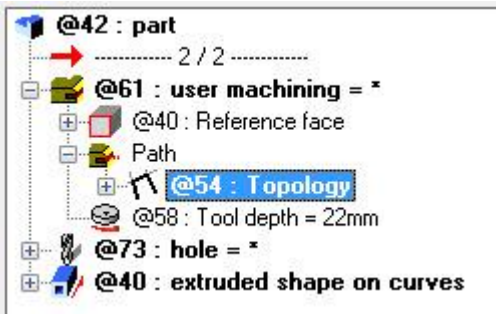


User machining as component process

From now is possible to define a user machining as a component process. The user machining allows defining the tool path to use to create machining operation.

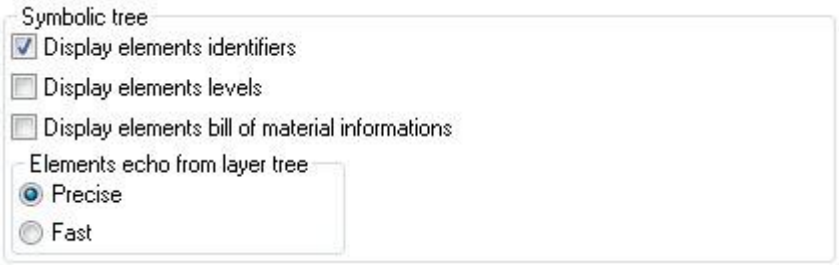
In this example of the hardware, the process is not groove or pocket.

So this user machining is defined in the hardware component with a special tool path.

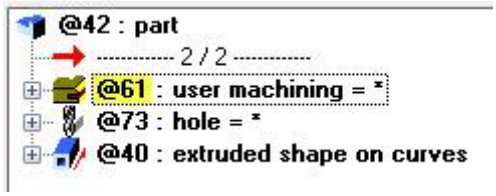


Once the user machining is created you can define it as tools.

Note: The user machining is not a physical operation. It is necessary to input it with its identification. To do this, you should write the identification from **Tools | Options |Others| Display elements identifiers**



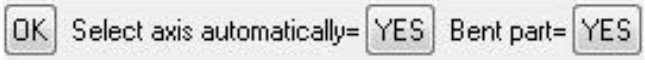
Define user machining with **Assemblage | Define component | Define tools.**
Like define normal tools, just give the name, when you need to select the local operation to insert, just write the identification of the user machining, you can get it from construction tree.



The user machining now defined as the component tools when you use this component and use automatically process.

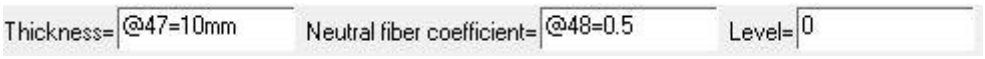
Wood unbent function

The new **Ben part** option allows unfolding the bend parts. The result of unfold part provides the unwound shape with all the characteristics of this part. These characteristics get the additional information (Operations, Dimension, Designation, Matter…).
This option is available when you define your parts. After you select the part for definition, just choose **Yes** for **Bend Part**.

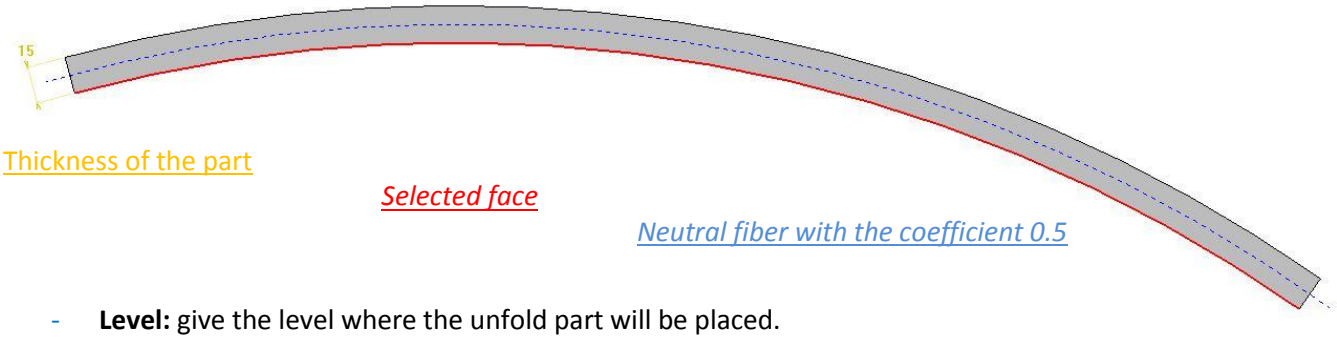


Note: This option is already available in the last version, but it allows only calculating the rectangle dimension refer to the selected face.

After selecting the **OK** button, the unwound options will be shown.



- **Thickness:** Here you have to give the thickness.
- **Neutral fiber coefficient:** Allows managing the distance between the neutral fiber and the unfold face with the input value. The neutral fiber is the curves who have the same length on unfold flat part. With the coefficient 0.5, the neutral fiber will be placed on the middle of the parts, 1 will be the other side of the part, and 0 will be the selected face.



Thickness of the part

Selected face

Neutral fiber with the coefficient 0.5

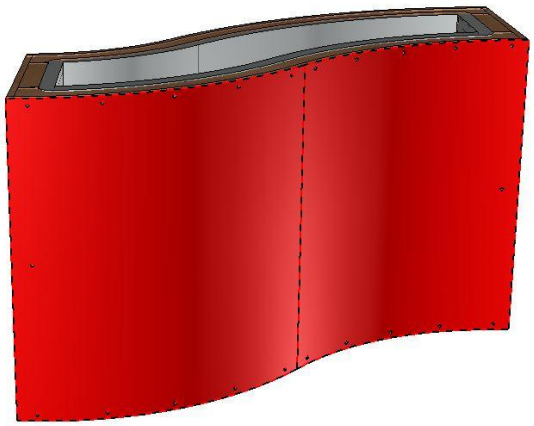
- **Level:** give the level where the unfold part will be placed.

- **Follow the tangent faces Yes/No:** Allows selecting automatically the tangent faces of the unwind face when you use the option Bend.

- **>>** : allow opening the advanced configuration parameter:

  - **Isoperimetric number**: Define the line quantity that will be placed on the Bend part to calculate the Unfolding. Higher this value is, better the quality will be but the calculation time will be long. The value 20 allows getting the good quality and a reasonable calculation time.
  - **Tolerance:** The tolerance allows defining the calculation of the geometries precision of you part. Smaller of the tolerance is, more better the calculation you can get, but you need long time for calculation.

- **Drilling for reference face Yes/No:** Allows getting the drilling on the reference face or not.



Warning: The angle of the unfolding drilling will be the same as the angle between selected face and the drilling. So to get the drilling vertical to the unfolded face, you should have a drilling vertical to the reference face.

- **Select the faces to unwind:** Allows selecting the reference face to do the unfolding.

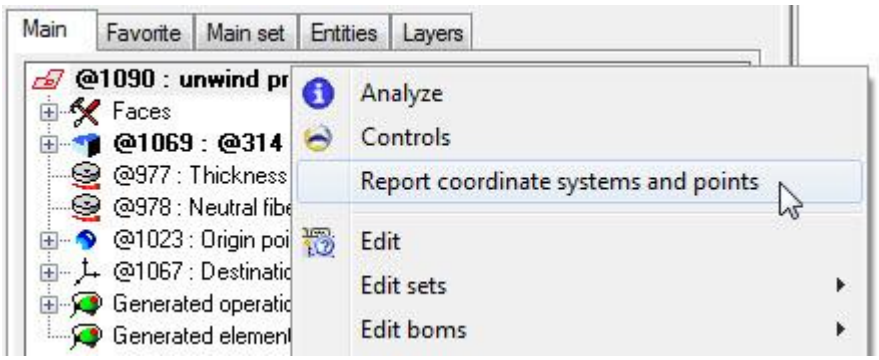
- Validate by clicking on **OK.**

- Then select the original coordinate system to do the unfolding. It is possible to select the axe that appeared on the point fund on the unfolding face. These axes are tangent to the selected face. It allows place the unwound part on the destination coordinate system.
- Choose the destination coordinate system for unfolding part.

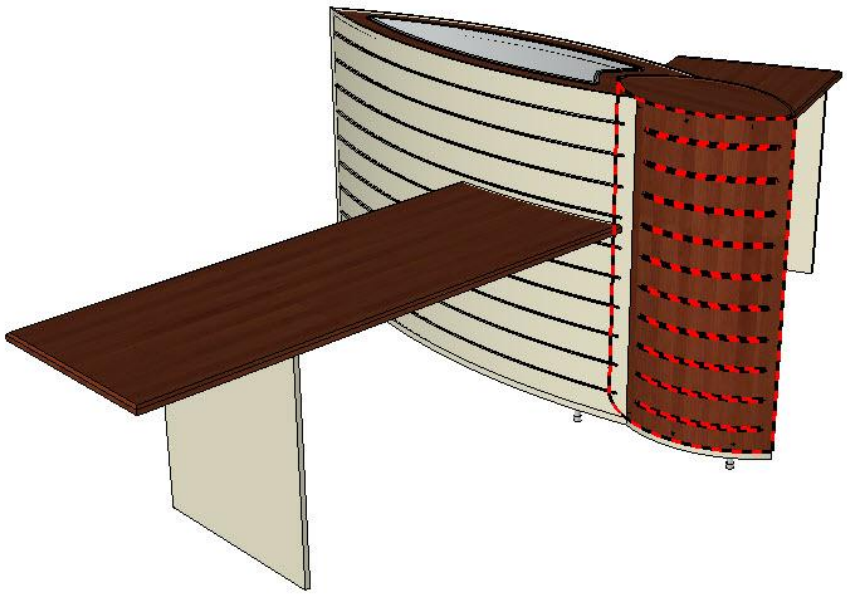
TopSolid'Wood creates automatically the unwound part and opens the window of the definition of the part.

Once the part is unfolded; it is possible, by editing the construction tree to modify several parameters.

It is also possible to right-click on the **Unwind processes | Report coordinate system and points** to select some additional elements (Frame or points) on the original part to add them to the unfolded result. It allows adding the machining operation point or frame to the final unfolded result to do the machining.



Project has the groove and the drilling operation on the original part



Unfolded part has the drilling and the start/finish points allow doing the machining on the unfolded part.

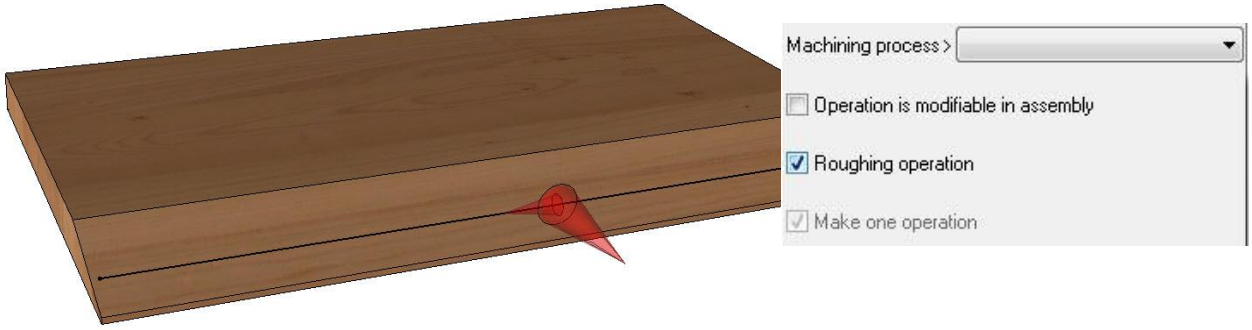
The **Bent part** option should always be enabled in place at the last level of assembly. For components with bent parts, it is necessary to define there with the **Bent part** option on **No**. It is in the final assembly, we have to redefine the bent parts selecting by the detection and choosing the **Add bend sawing-up** option.

Roughing operation

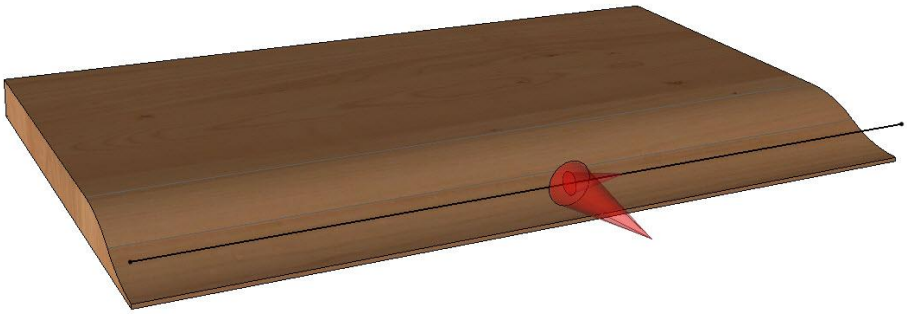
Now it is possible to define the wood operation (**Mounding, Groove and Rabbet**) as the roughing operation.

This one allows defining the machining with two paths with two different tools, and be able to do two machining steps in TopSolid'WoodCam.

- Create the first Moulding machining with the Roughing tool and select the option **Roughing operation** in the configuration windows.

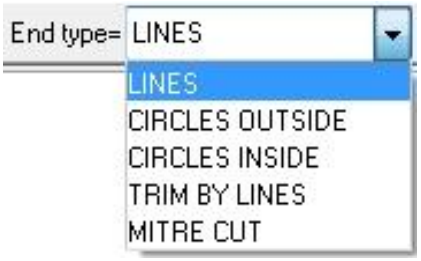


- Create the second Moulding for finishing operation.



Limitation of the ends of thicken profile

The two new options are available to the operation **Thicken** to limit his extremities refer to other profile, by the mode miter cut or planar cut. This option is very useful to manage the cutting between two parts created by trace thicken.

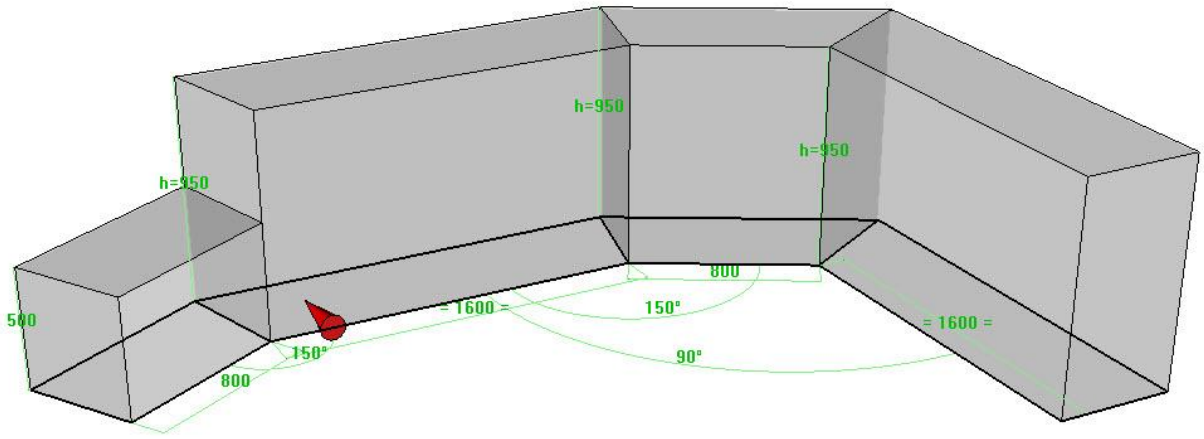


These new operation are available in the rolling list **End type** when you thick the profile.

- **Trim by line:** Allows limiting two ends by selected lines.
- **Mitre cut:** The ends of the profile will be the middle angle line between reference line and the two profiles selected.

Once the reference profile selected, 3 options allow doing following modification:

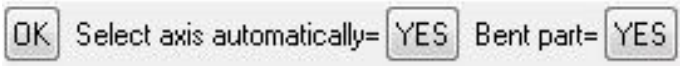
- The option **AUTO** detected automatically the extremity line and limited Thicken line.
- The option **ALL SEGMENTS** detected automatically all the components of the contour and create all the thicken component on all the segments.
- The options **nothing** allows not limiting the profile on one side or on the other side.



This Extrude Block is a reception desk comes from the thicken line by miter cutting mode.

Improvements of part definition

When you define the part, the cutting-up frame is now linked to the selected axes. Use the associative frame allows getting the automatic debit update by the dimension and the position of the parts.

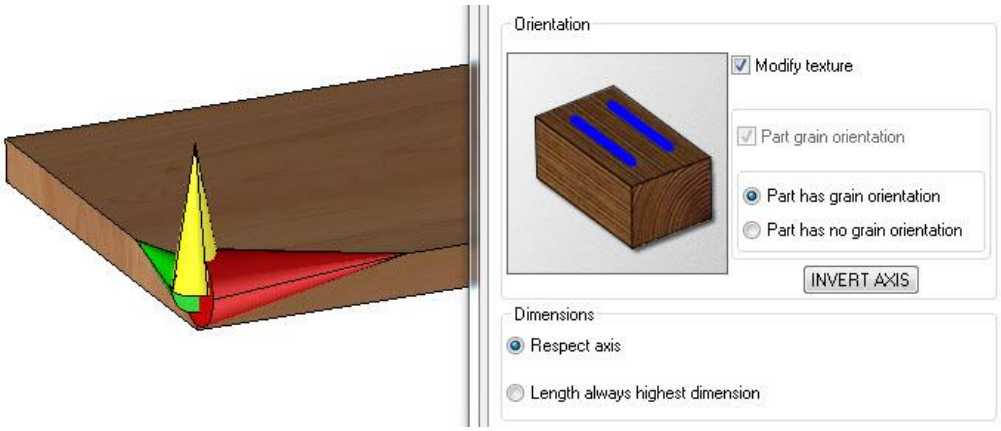


A new way to use the part definition

To continue to use the frame of cutting-up non-associative of the selected axe, you should use the configuration mode *D_PROP_ZWOO_SAWINGUP_ASK_ASSOC_FRAME* with the value 1, in the file *topzwoo.cfg* placed in the folder *Missler\Config\V614*. The function of part definition will be changed, when you define the part, it will ask you the frame is associative or not.

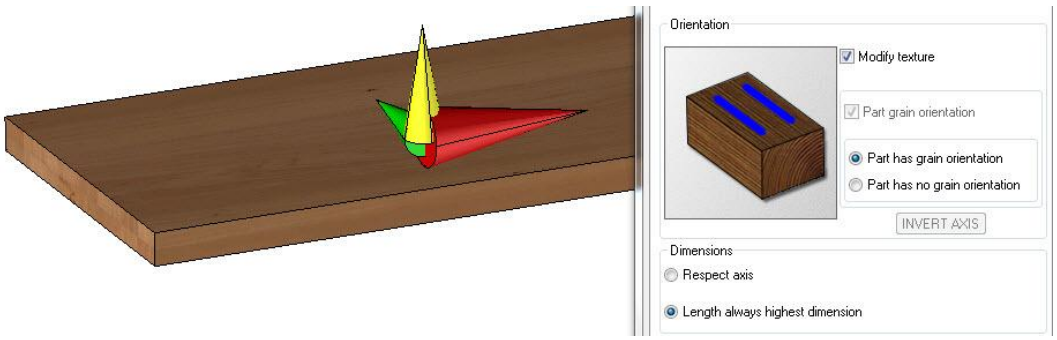
In the **Cutting-up** tab, when the part is defined with the mode **Select the axes automatically** :

- The mode **Respect the axes** is used by default with a frame associative link to part edge. In this mode, It is possible to inverse the axes by clicking on the arrow that placed on the part. When we click on the yellow arrow, we can turn over the frame with 90° (It is the same result when we click **Invert axes**).



- If we cannot find the cutting-up axes automatically, an error massage will be shown on the Alpha bar (**Axes not found**) and you must define axes manually (if the **Add to cutting-up** button is selected, otherwise, the axes will be selected when the part is added to cutting-up).

- If the mode **Length always highest dimension** is used, the associative frame created will be the position with the X dimension in the longest dimension of your part.
  In this mode, the button **Invert axes** is not usable and it is not possible to select the yellow arrow.



- If we use the mode **Respect axes** and the axes of the part is not fund automatically, you will have an error massage in the alpha bar and the definition window is hidden to allow selecting the axes for definition.
- If all the modification of axe of debit, and the over dimension don't change between with and the length, it means over dimension on length is always on length, whatever about the axe direction.

Note: In the mode **Edge**, if the axe is invert, the over dimension will changed automatically to follow the width and length of the panel.

Sizes	Values	Modes	Over dime...
Length	400.0mm	edge shape	0mm
Width	196.0mm	edge shape	4mm
Thickness	19.0mm	additional	0mm

Sizes	Values	Modes	Over dime...
Length	196.0mm	edge shape	4mm
Width	400.0mm	edge shape	0mm
Thickness	19.0mm	additional	0mm

Improvement of the **Add to cutting-up** button:

- The situation of Cutting-up is link to each part: one part didn't add to cutting-up is always no cutting-up, until you select cutting-up.
- When you define the part not yet define, the Adding-up option will keep the situation that you used last time.
- When you define the part already defined in the old version with TopSolid'Wood 2013 :
    - If the part does not have information about debit, **Add cutting-up** will like a new part not yet defined.
    - If the part has a frame of Cutting-up: the **Add Cutting-up** will be selected automatically.
    - If the part have the machining and draft property, but not the Cutting up information, the **Add Cutting-up** will not be selected.

- If the part definition is not added to cutting-up, the window of definition will open automatically, without the dialogue bar.
- When you define the part, if you select several parts and click on stop, all the parts will be defined automatically with the associative Frame in mode **Respect axes**, and they will be added to cutting-up.

Improvements of Define multiple parts

From today, if the selected part of multiple definitions is not yet added to cutting-up, they will be added to cutting-up automatically, in the mode **Respect axes**.
If the axe is not fund automatically, the message will be shown in the Alpha bar and part will be grey in the multiple definition windows, and it is not possible to modify the part no added cutting-up with multiple definition.

For avoid the error axe of cutting-up in multiple definition, it is not possible to invert the axe direction and use the mode **Respect axes** or **Length always highest dimension** in the **Define multiple parts**.

Define the part on the repeated parts:

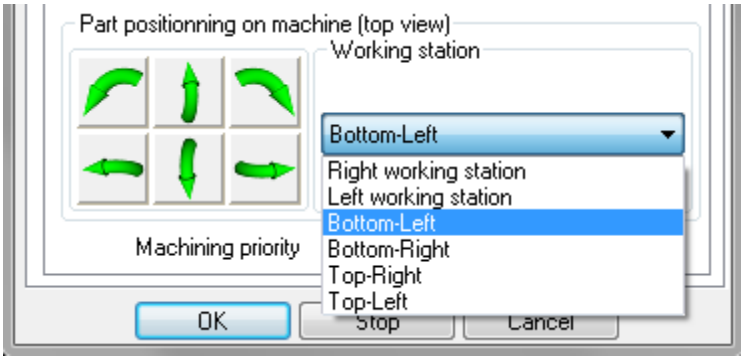
- If all the assembly is selected, the BOM will show all the parts de repetition: the modification is local for each part.
- If the repetition is selected by clicking on the parts. The BOM will consider the template of the repetition: the modification is global for all the parts of repetition. The modification will not rewrite the information of each part, if the part is modified locally.

Configuration of machining workstation

Now it is possible to predefine the machining working station in **Tools | Options.** You can define one list of working-station for multi machining. This rolling listed is available when you define the part in the machining Tap.

Note: The 2 default working station in old version is proposed in the rolling list to have the compatibility with old version files.

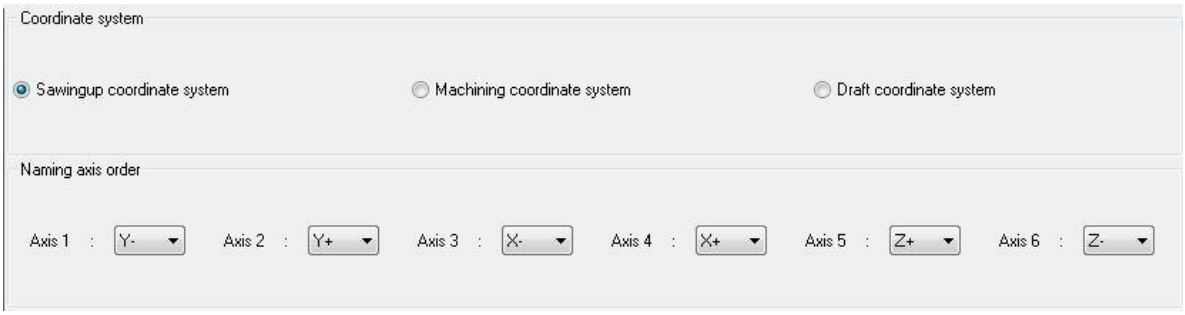
Name	Designation	Work station 1	Work station 2
Bottom-Left	Bottom-Left	X	-
Bottom-Right	Bottom-Right	-	X
Top-Right	Top-Right	-	-
Top-Left	Top-Left	-	-



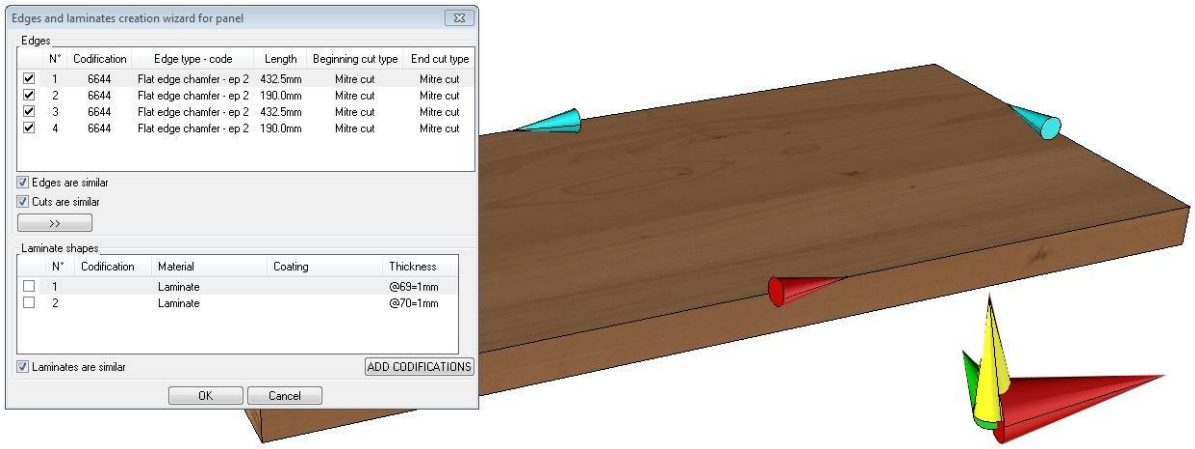
Other improvements of panel entity

When creating the panel

When you create or copy the panel, all the edge will be positioned refer to the coordinate system and the first axe selected in **Tools | Options | Configuration TopSolid'Wood | Edge/Laminate**. All the edges are oriented on the part like the codification in the BOM.



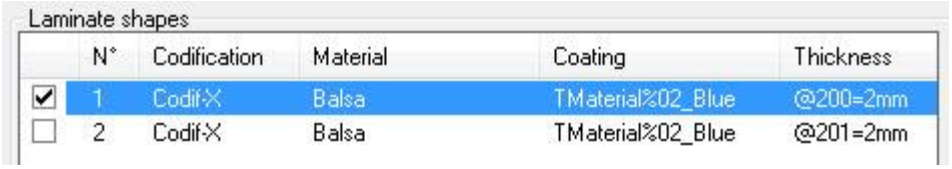
The first edge will be always on the axe 1 refer to the selected coordinate system on the part. (Cutting-up coordinate, machining or Draft).



Here the edges are positioned refer to cutting-up coordinate system, 1st edge on Y-.

Then, the other edges are positioned in the rotation order of Z+ from the selected frame, like the yellow arrow in this example.

In the edges creation windows, we can select or unselect the laminate for add or delete them.



When copying the panel

Attribute and property of the panels

In the document TopSolid'Wood, if the option copy panel is used to recreate the panel entity from some panel already existed, the edges, the laminate, and the options of the panel already existed will be copied. No property will be copied (designation, material, type…).

If the panel is copied from the **panel template**, all the properties and attributes will be copied on the new panel.

Note: The material and the coating will not be copied from the **panel Template.** To copy the material and the coating of the panel template, you have to copy the following configuration words *ZWOO_D_PROCESS_PANEL_COPY_MATTER_COATING* with the value 1.

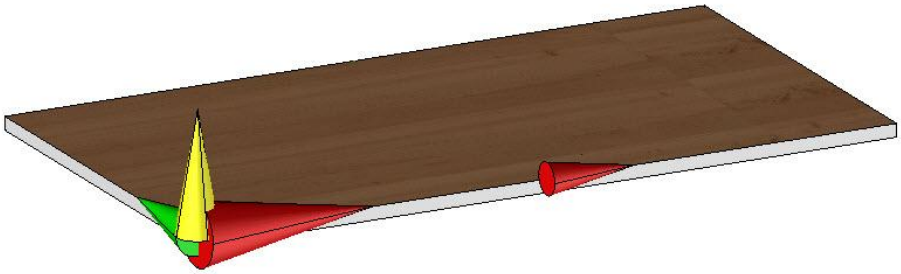
Defining the part on the panel not yet defined

When you copy the panel on some part not yet defined, TopSolid'Wood will link some default cutting-up property to part (on associative frame) without open the part definition window. We can change this mode by put some configuration value *ZWOO_D_PROCESS_PANEL_COPY_OPEN_DEFPART* with value 1:

- If the config value is 1, **Define part** window will be opened.
- If the config value is 0, **Define part** window will not be opened.

Then, the cutting-up frame will be shown on the part and one red arrow on the first edge.

Here is possible to modify the frame by clicking on the arrow. If the edges are positioned refer to the frame, I will be changed automatically after changing the frame.

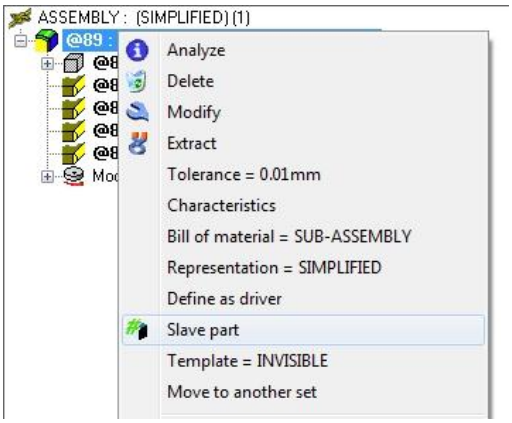


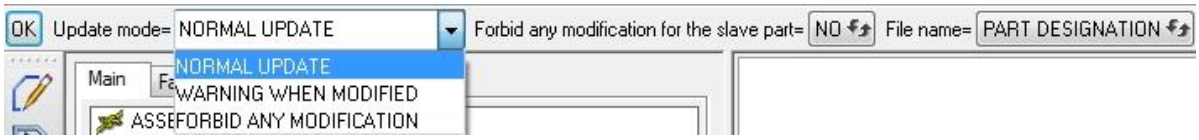
The red arrow allows changing the direction of the edge.

Slave panel management

With the version 6.14, you can manage slave panel entity. The method to do this is the same as the slave part from just one part.

- In the construction tree, **right-click on the entity panel | Slave part**.



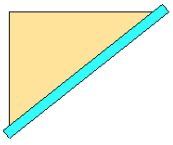
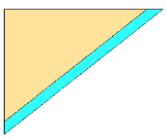
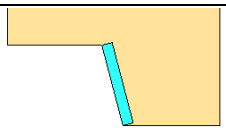
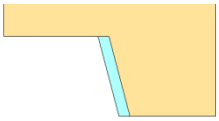
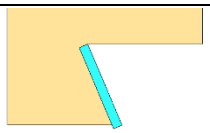
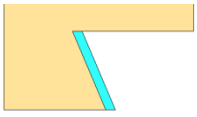


- **Normal update of the original part**:
  - **Normal update**: The slave part will be updated automatically.
  - **Warning when modified**: The slave part will be updated with warning to user.
  - **Forbid any modification**: The original part cannot be modified.

- **Forbid any modification for the slave part**: Forbid or not the modification of slave part.
- **Name of the file**: Allows you to change the name of the file of the part.

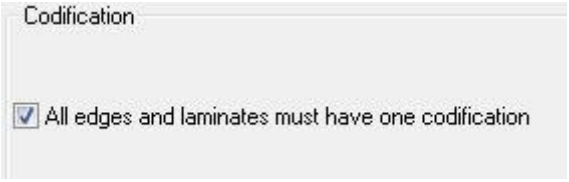
Improvements of the end cut types of edges

For the panel function, the limitation of the end cut types of edges will be better in some special case:

Result in version 2012	Result in version 2013
	
	
	

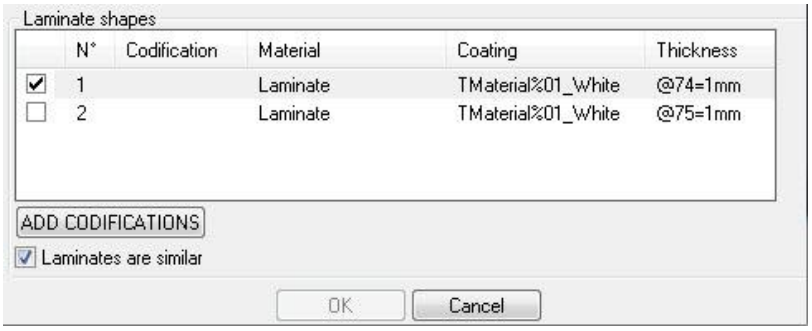
Edge codification improvements

Several improvements in this version are be done for edges and laminates management.
A new option **All edges and laminates must have one codification** is available in **Tools | Options | TopSolid'Wood Configuration | Edge/ Laminate | Codification**.
If this one is selected, the temporary edge and laminate cannot be valid; they must be linked to some codification.

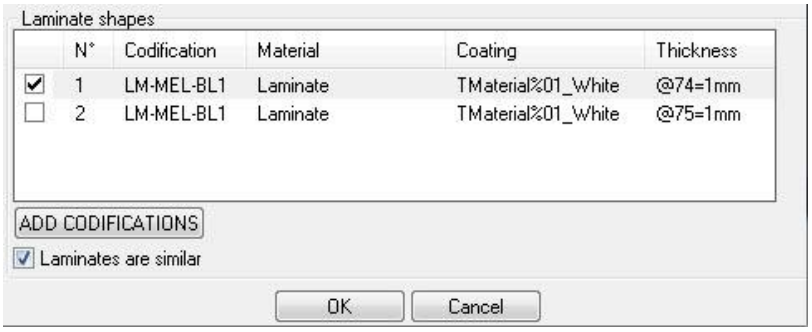


So, if this option is selected, when you create the panel if there is no codification linked to the Edge and the melamine surface you cannot validate it.

For example, in this Panel configuration window, the laminate is not linked to some codification. And the OK button is grey, you cannot use it.



The **Add Codification** button is useful to directly create the codification of laminate.

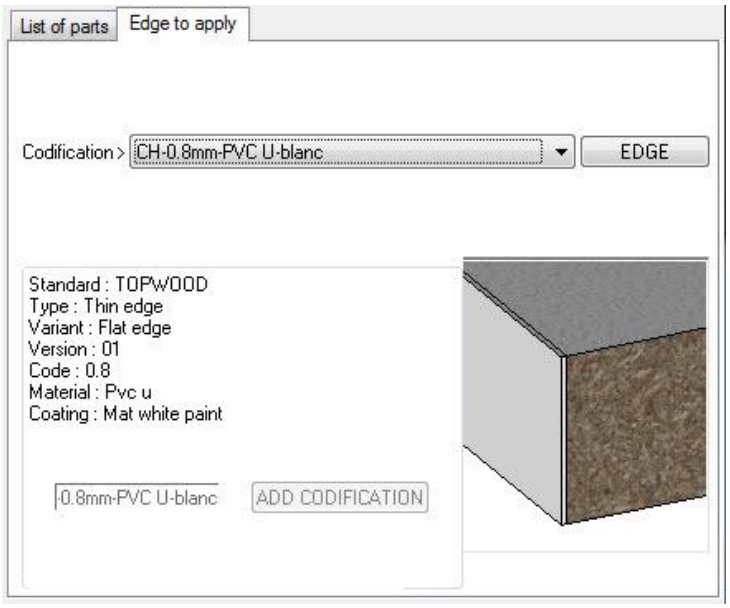


Once you created the laminate codification. It is possible to click on OK to validate the panel configuration window. The codification that you created is added to the TopSolid'Wood library.

This operation now is available in the function: **Edge, Laminate, Panel, Edge configuration, Laminate configuration.**

Now, in order to avoid use some material no prepared for the edge and Laminate, the edge and the laminate is no more available in the material configuration function.

Edge configuration: In the **Edge to apply**, the codification of the edge is shown.

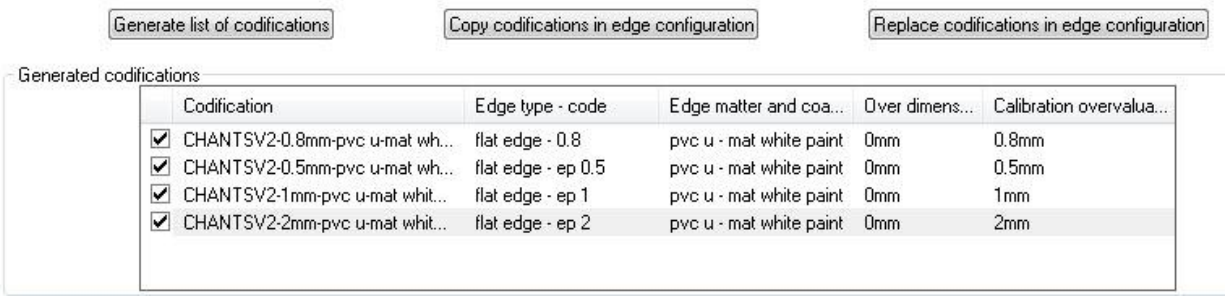


Improvement of the Edge codification automatic generation

In **Tools | Options | TopSolid'Wood Configuration | Edge / Laminate | Automatic codification of Edge,** it is possible to generate automatically the codification of the edge by some rules. This codification can be added to the Edge codification list by clicking on the Button **Copy codifications in the Edge configuration.**

With this option, if this codification is already created, an error massage will be shown and it is not possible to copy the Edge. From now, the button **Replace codification in the Edge configuration** allows you to automatically replace the existing codifications.

This function is very useful in case you have new materials to use and/or the codification rules have changed.

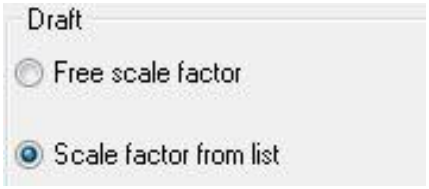


Improvements of multi draft

In the version 2013, several points of Multi Draft are modified in TopSolid'Wood.

Free scale factor

Now it is possible to select if the scale factor of the multi Draft is free or link to the factor in the list. You can make this one in the **document property or the application property.**

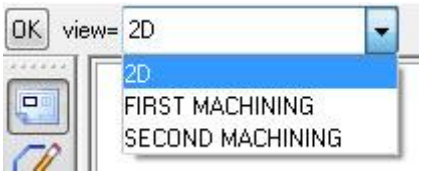


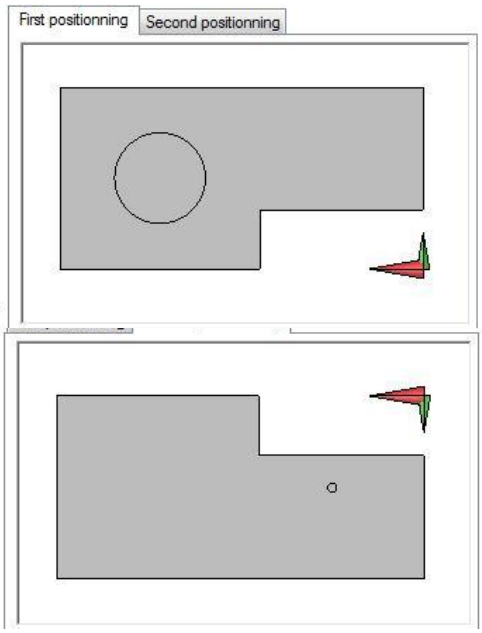
When you use the multi Draft function, the function will automatically calculate the dimension of the 2D view to adapt the dimension of the Draft document:

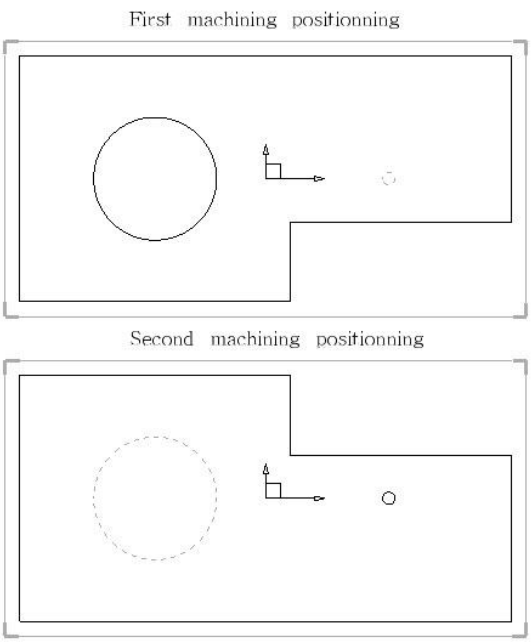
- If the scale factor is free, it will find the correct dimension for draft.
- If the scale factor is from list, TopSolid'Wood will automatically select automatically the nearest smaller value than the automatic one in the list.

Machining View in the template

Now is possible to put the machining view in the template. You need to use **Define part | Machining | First position/Second position.** To position the view you need to use the function **WOOD | Multi Draft |Create template | Create view | First Machining/Second Machining.**





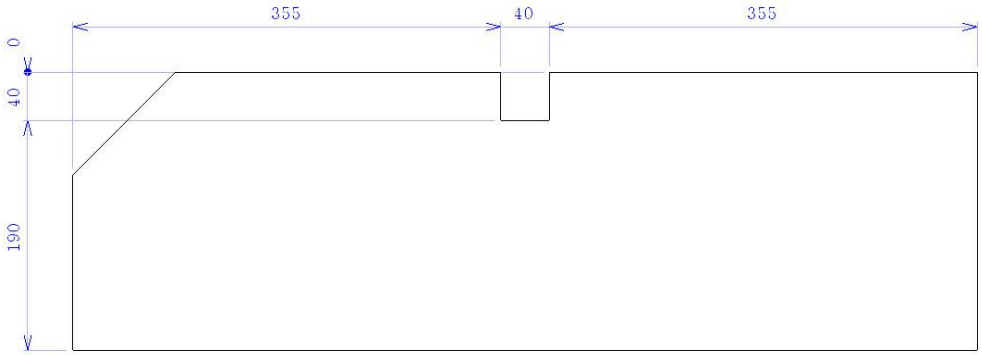


Scale factor of 2D view: When positioning the 2D view, we propose you to use the factor 1. In fact, when you use the multiple drafts, the software wills calculate automatically the best scale factor to position the view. If the factor is different than 1, it will multiple them with the best factor.

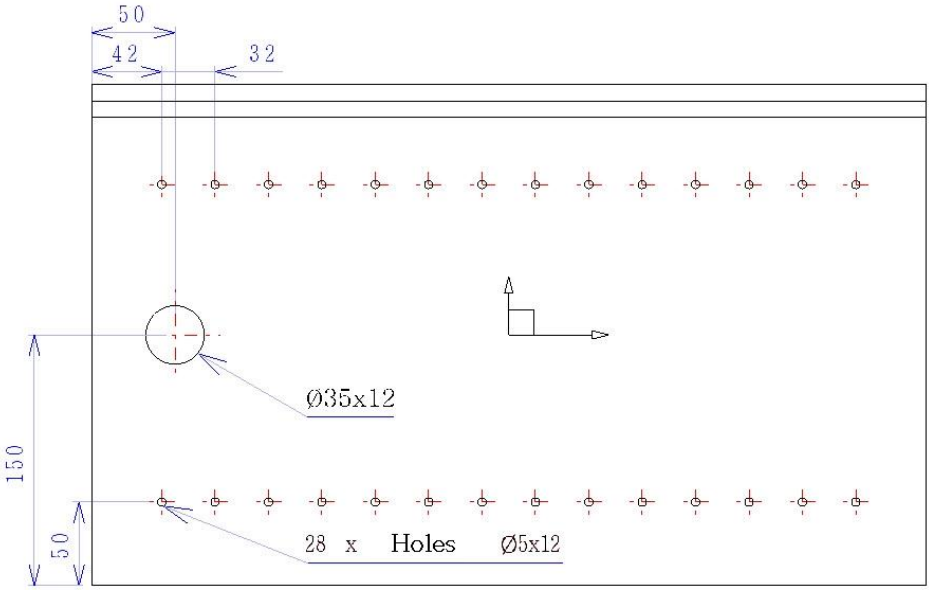
Example: If the factor 0.1 is used here, and the function Multi-Drat find the best factor is 0.5, so the result of factor of 2 D view will be 0.05.

Various Improvements of use

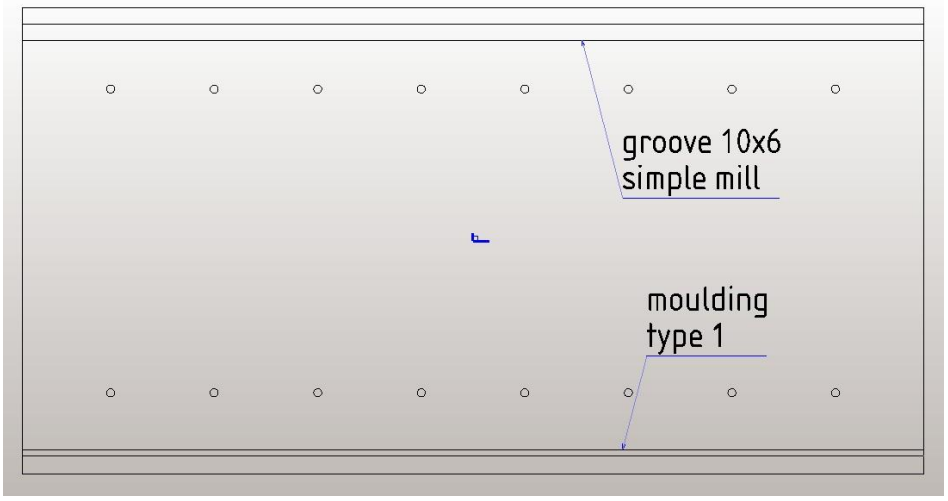
- The dimensions of the part are created by composite dimension.



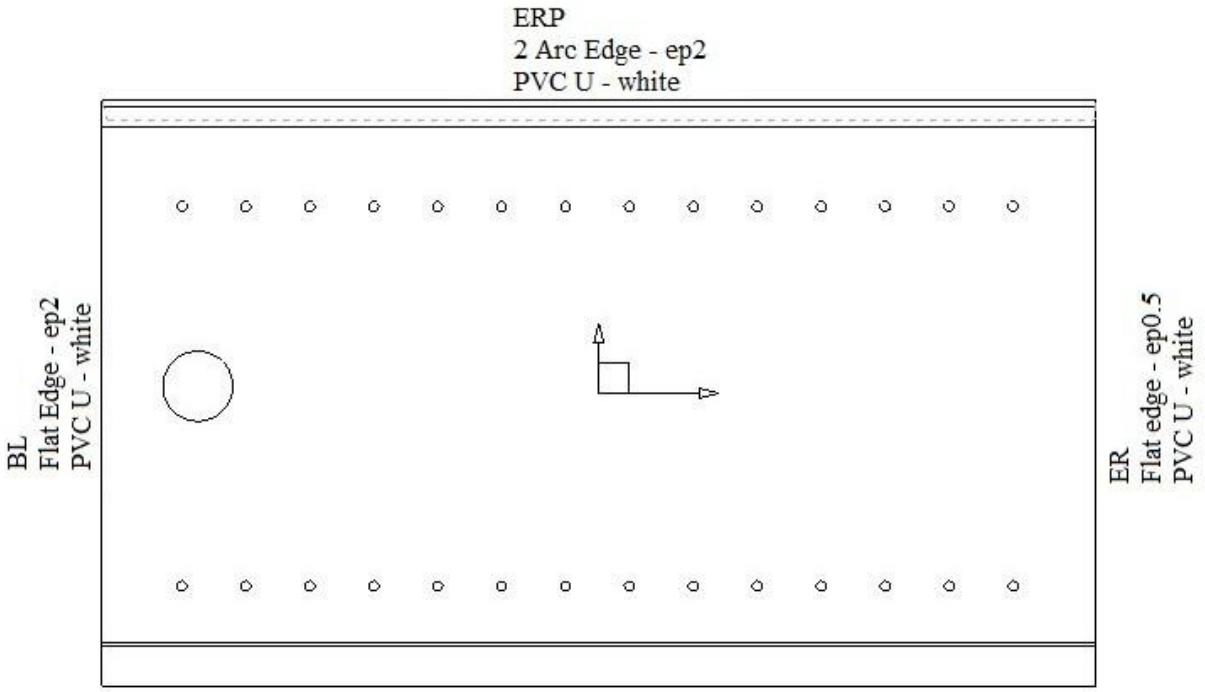
- The drilling dimension is now supported on the draft.



- The dimension of wood operation: the note created on the wood operation is now composed by 2 lines; the first line has the name of operation and second line has the machining tools information.



- The edge dimensions are shown in the direction of the Edges and will be on the center of them.



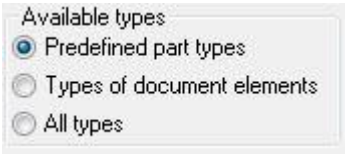
To manager better the view and avoid overcharge of the draft, the automatic dimension will be only shown on the principal view of the draft document.

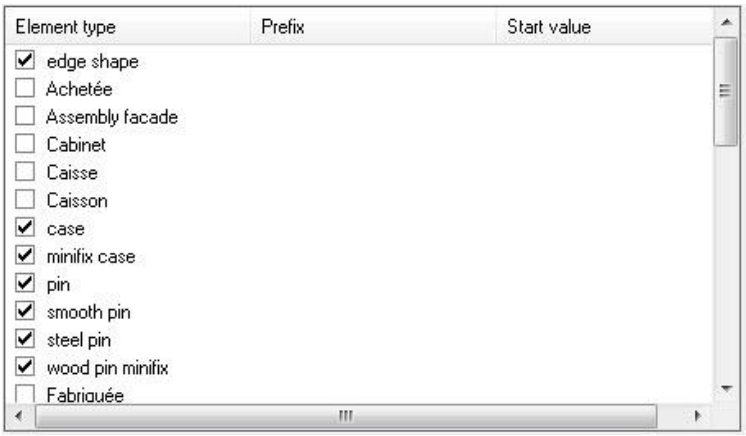
If you want to use the old way show all the edge dimension, you can use the configuration *ZWOO_DFT_NEW_EDGE_NOTE* with the value 0.

Improvement of the 3D index

Options setting

In **Tools | Option | Predefine index**, the type list is now shown either by **Predefined part types** or **Types of document elements,** or both with **All types.** The types of parts you don't want to number must be selected.



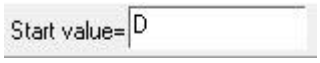


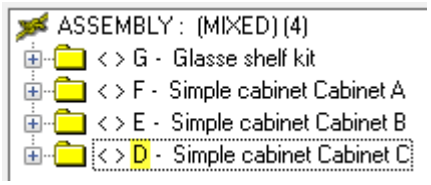
This list allows you to select the types of parts for which we do not want to predefine the index. We can prefix the number for each type of part and give them the start value for several types.

Start value for automatic index

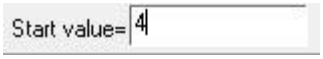
For the automatic index, it is now possible to define the start value of the first level.

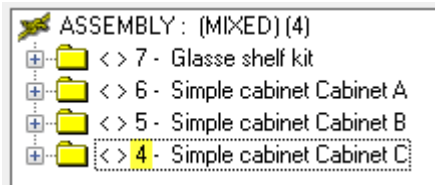
- If the automatic index starts with a letter, a letter is required.





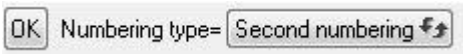
- If the automatic index starts with a number, a number is required.





Second numbering for parts

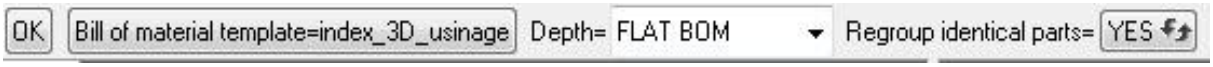
A second type of numbering, the **second numbering**, is available with the automatic numbering of elements.

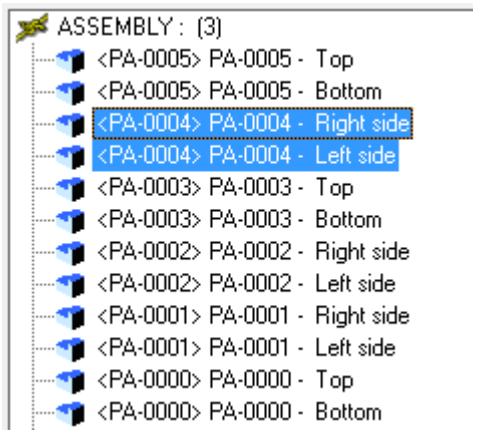


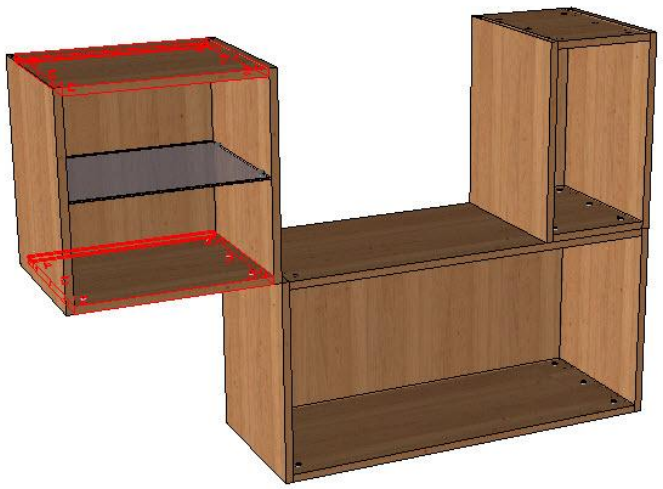
Possibility of using the principal numbering or second numbering for the Automatic index. The two types of automatic numbering work the same way as in previous versions.

<u>Example to use the two type of numbering</u>

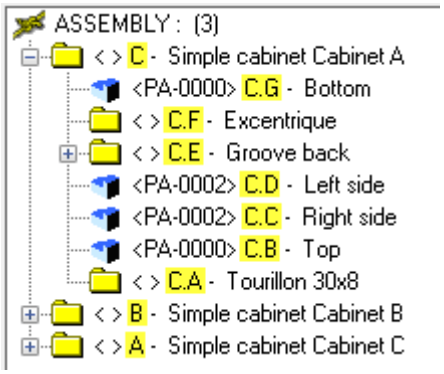
- **Principle numbering**: Allows you for example to provide the same index for identical geometries to manage only one program for several parts.

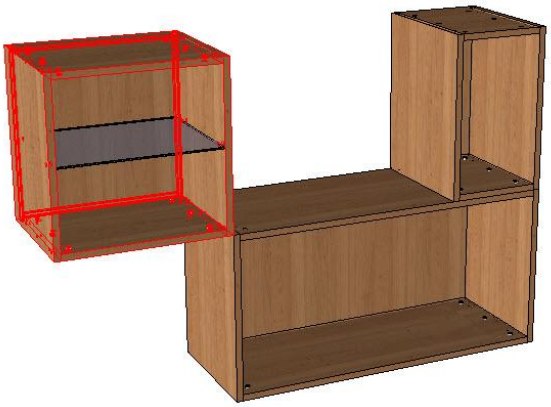






- **Second numbering**: Allows you for example to create the index in the multi-level mode in order to identify the parts in the different subassemblies of the project (see examples shown in yellow below).





Ardis Interface

The Ardis interface has been improved to allow you to export the machining information to Ardis. The interface of previous versions with the format R41 is always available, but you can get a new format XML (you can manage it in **Tools | Option | TopSolid'Wood Configuration | Cutting off |Ardis Configuration**) that have the machining information for each part. The following machining operations are supported:

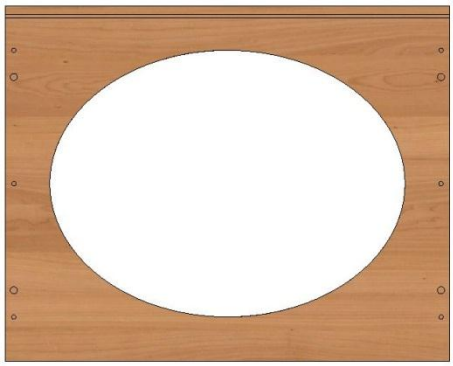
- Calibration;
- Drilling/multi-drilling;
- Groove/Rabbet;
- Pocket;
- Moulding;
- User machining

All setting options are created in **Tools | Option**.
For more information about the Ardis XML interface, refer to the *Ardis XML Interface* training document.

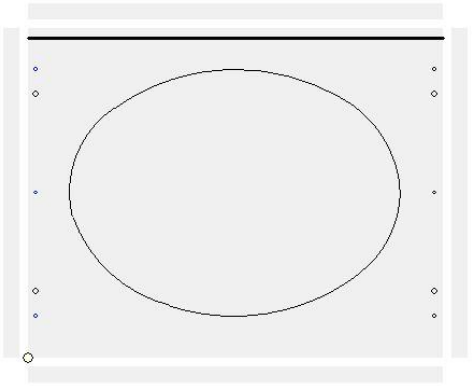
Warning: In order to use this export, some special additional tools are required in Ardis. For more details, refer to the *Ardis XML Interface* training document.

Example: TopSolid'Wood exports one part with the Groove, drilling and pocket information.

↑ *TopSolid'Wood Project*



Part in TopSolid'Wood

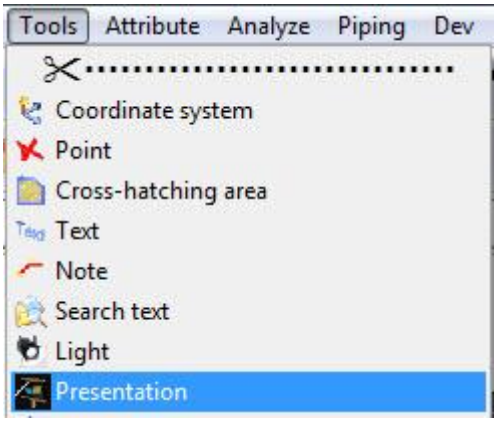


Part in Ardis

	Function	X	Y	Length	Width	Radius	Z2	Side	Operation
▸	DRILL	490.50000	50.00000			2.50000	8.50000	Back face	
	DRILL	490.50000	80.00000			4.00000	10.00000	Back face	
	DRILL	490.50000	320.00000			4.00000	10.00000	Back face	
	LINE	0.00000	388.00000	500.00000	388.00000		11.00000	Back face	Groove
	ARC	450.00000	200.00000	374.69796	317.27472	139.17864	19.00000	Back face	Pocket internal
	ARC	374.69796	317.27472	103.52833	302.13806	250.46515	19.00000	Back face	Pocket internal
	ARC	103.52833	302.13806	54.39299	168.74070	124.76142	19.00000	Back face	Pocket internal

↑ *Ardis information of part exported from TopSolid'Wood.*

Document presentation

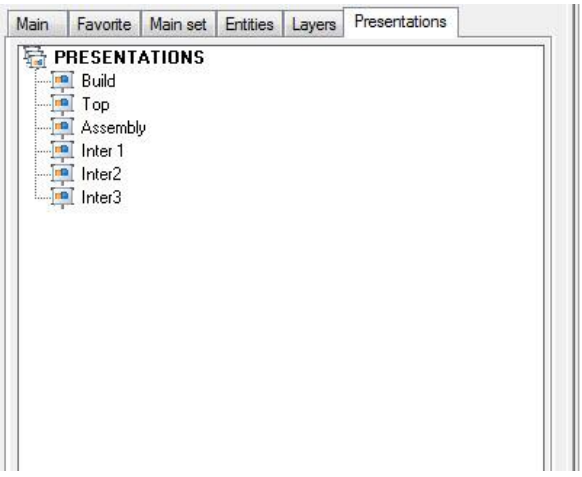


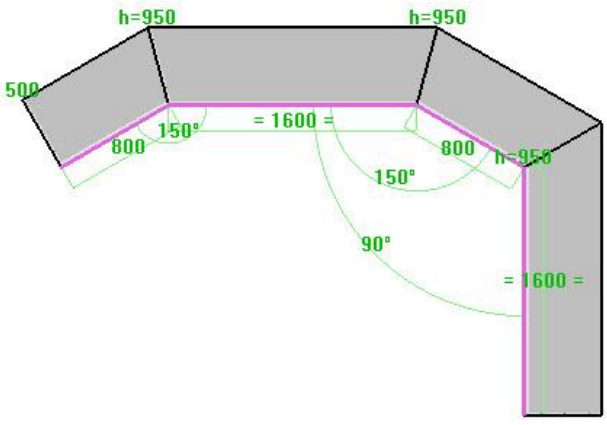
The new function in **Tools | Option | Presentation** allows you to save the presentation of a Design, Draft or Machining document.

These presentation save all the graphic information (orientation, rendering mode, zoom, multi-windows, graphic cut, act if level, hidden element) of your document.

The presentations thus created are automatically included in the new **Presentations** tab of the symbolic tree in which a contextual menu allows you to:

- Add the presentation
- Rename the presentation
- Show the presentation (a presentation can be displayed just by dragging it into the graphical area).
- Delete the presentation
- Redefine the presentation
- Calculate the rendering for each presentation (if you have TopSolid'Image). This new option allows you to calculate different images for different presentations saved. These images will be renamed the same way as the presentation.





Presentation of the top view, with the profile and the dimension.

Presentation with perspective view, with separation level hidden.

Improvement of the background image

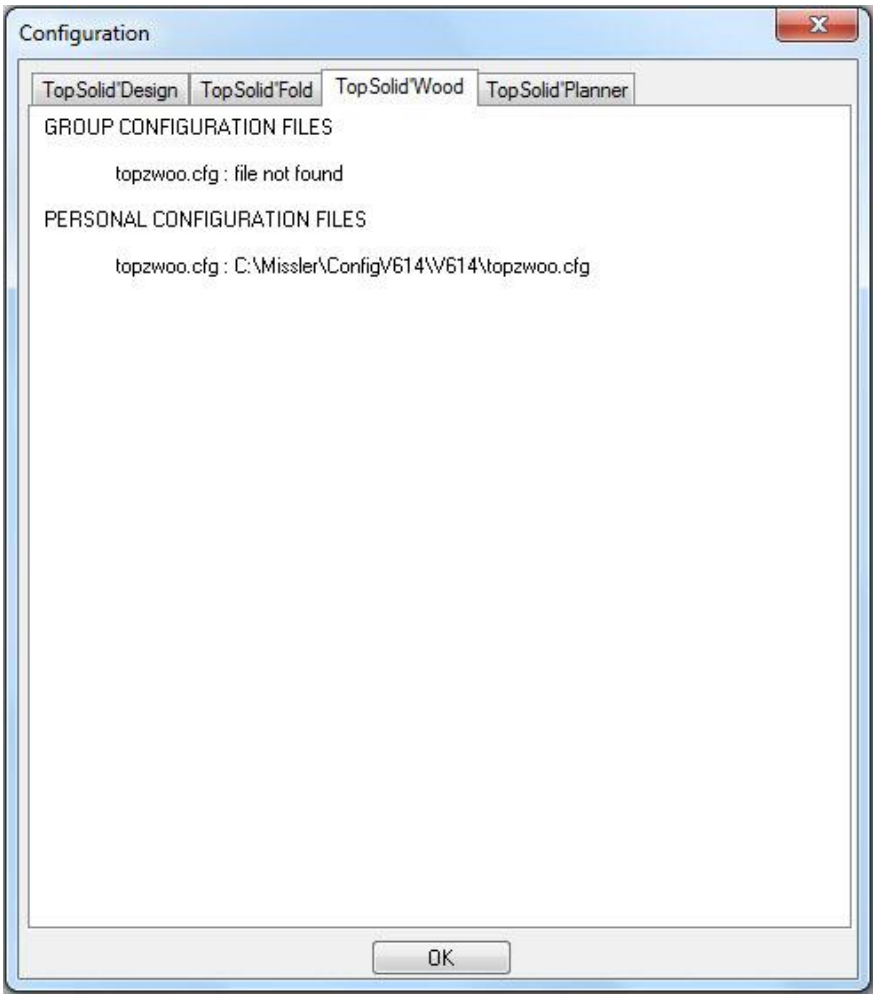
The **background image** is now available for assembly components.

You just have to insert the background image in the component. If the image is linked with the component dimension, it will follow the transformation of the component in the assembly (position and dimension). This improvement is very useful for decoration items.

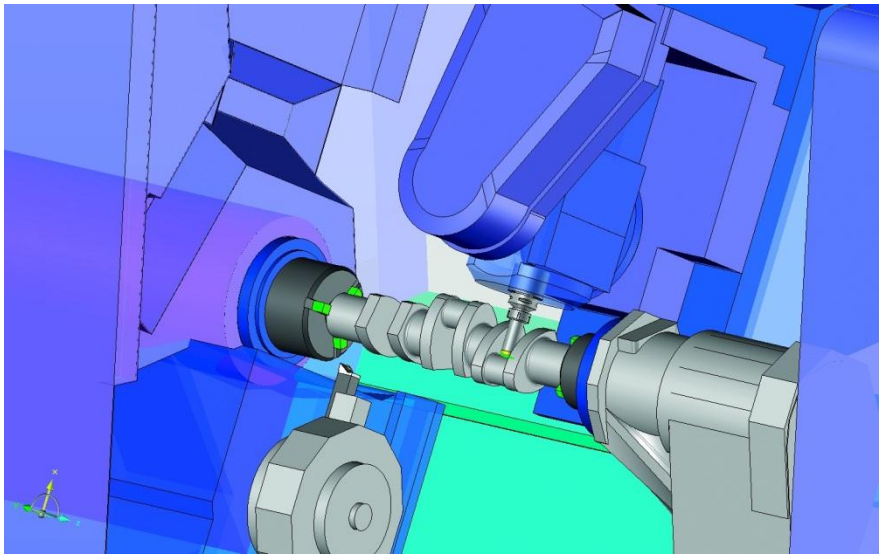
Configuration management

The function **Help | Configuration** has been improved to give you the indication of configuration management.

The applications are separated in different tabs and you can have more information about the local and group configuration setting.



TopSolid'Cam 2013: What's New



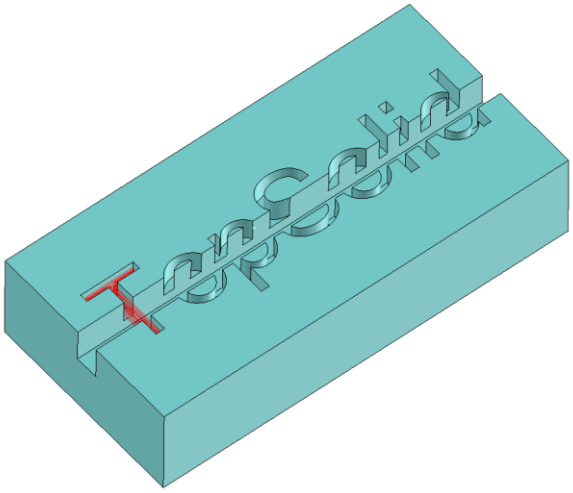
This document describes the improvements made to the **machining** application of **TopSolid'Cam software**: 2013 version.

2D Milling

In 2D milling operations, it is now possible to use the clearance block to manage retracts (at the end or between passes) between drilling, pocketing and contouring operations.

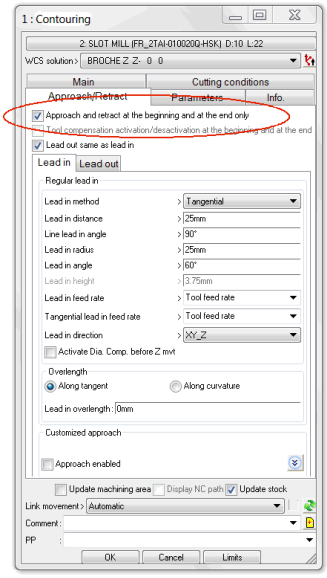
Feature recognition

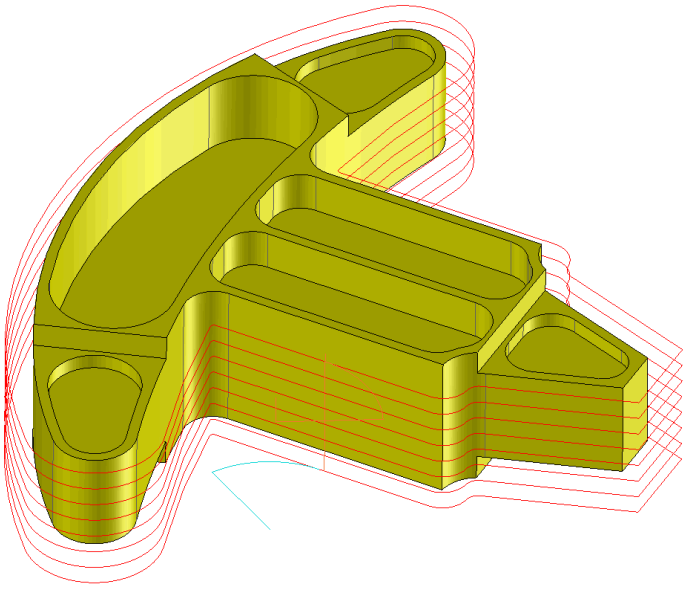
- Pockets and slots can be recognized, even if they have no bottom face, provided they have been created using the pocket and slot TopSolid'Design feature.



Contouring

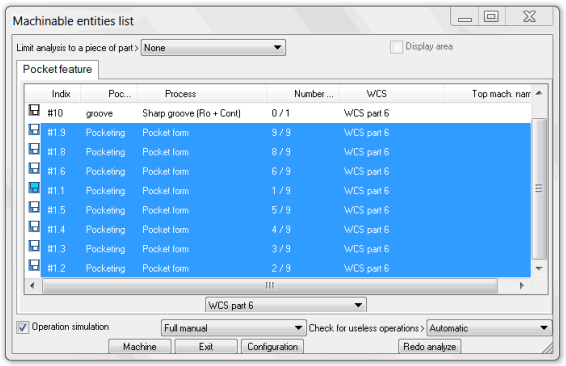
- It is now possible to add a fillet on right angles.
- Many improvements have been made for WoodCam use:
  - Customizable vertical plunge;
  - Approach at the beginning of the first pass and retract at the end of the last pass when there are many passes in Z;
  - ...

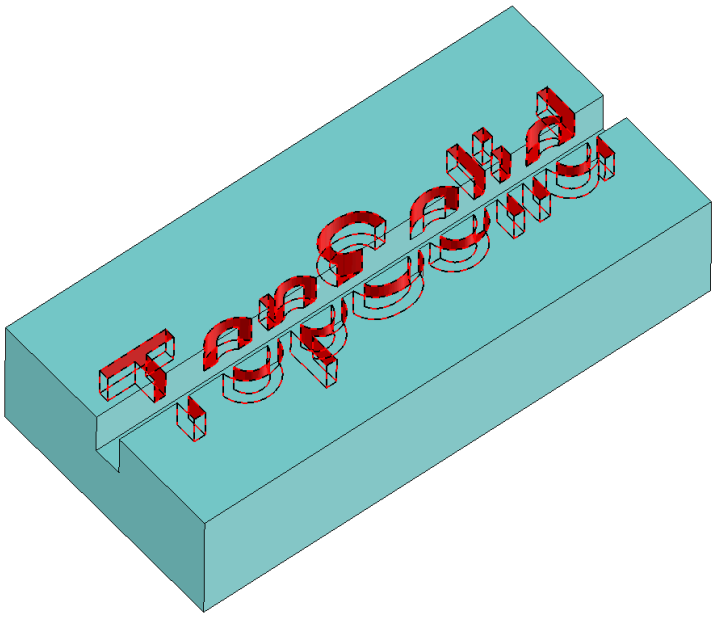




Pocketing

- When a pocket is cut in many portions (because there are some islands), it is now possible to stay on the pocket Z level and thus avoid jumps.
- TopSolid feature pockets based on multi-profiles are supported. The feature is shown as N machinable items.





Drilling

- A new setting allows another security distance to be defined to secure the jumps over the obstacles (clearance distance on collisions).
- Machine cycle for threading in WFL machines has been added.
- Collision checking is available when the user drills the pocket entry points. The problem occurred when the centering tool was bigger than the drilling tool.
- The Management of the intersecting holes has been completely rewritten. Nonetheless, the former version is still available via the **Option** menu.
- The ¾ drilling supports pecking and clearing options, even in case of intersecting holes.

3D Milling

Roughing

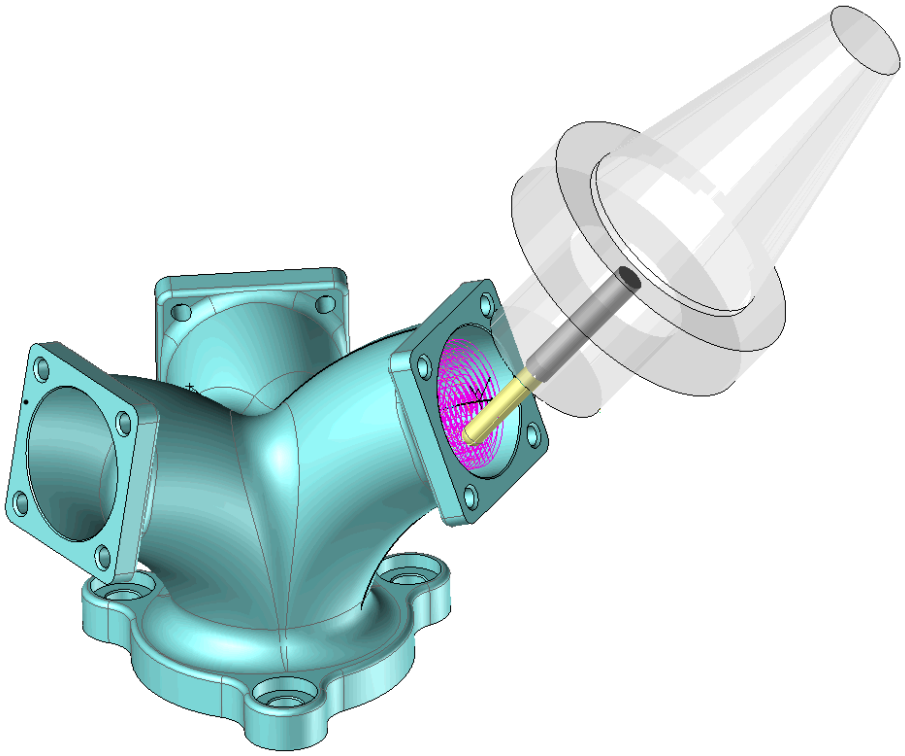
- The roughing calculation time in **pocket** mode has been improved. On complex parts, a factor 2 has been noticed.

Contouring by plunge

- In contouring by plunge, the **Corrected** and **Original** modes are supported.

Finishing

- PPT between two curves has been improved by the addition of synchronization parameters.
- New sweeping possibility to be able to machine admission pipes. Available in 5-axis also.



- All the limitation curves can be deleted in one click.

Super-finishing

- Possibility to use the curves limitation bounding mode (on, before, after).
- Collision checking with the tool holder has been added.

4D Milling

- Parameters, like angular split values, starting C values which were defined with configuration words, are now available in the dialog boxes.

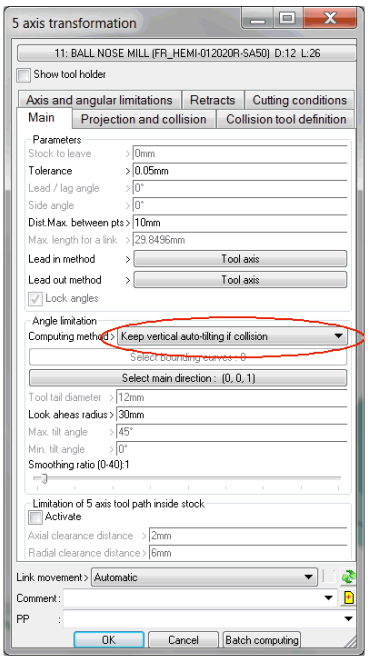
5D Milling

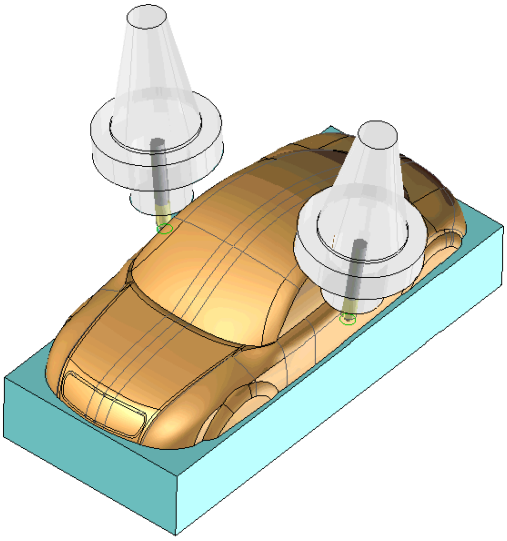
Swarf machining

- Better management of areas where the tool cannot go when its diameter is too big.
- Helical approach and retract have been added (also available in 5-axis contouring). Be careful, there is no collision control on these approaches and retracts.
- On specific cases, it is possible to lock the Y axis to ZERO. Be careful, in these cases, we are not sure to produce the right part because the normal vectors are modified! Conversion of a translation movement into a rotation.
- Approach and retract can be sliced according to the initial and final tangents of the curves to machine.
- Retract modes between passes and at the beginning and the end of the operation have been improved (also available for 5-axis contouring).

3D to 5D

- A new tilting mode keeps the tool vertically and tilts it to avoid collisions when they appear, and only when they appear.





Basic operation sets

- Each operation can be used with a no generate attribute.

Retract to the tool change point

- New parameters set in which coordinate system are the coordinates input by the user. This is useful to save and apply retracts to another file. "ctp_type_coord"

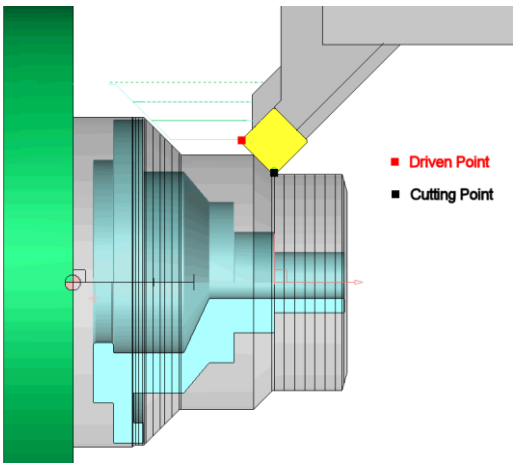
Turning

Parting off

- A new parameter sets the feed rate of the retraction to the retraction diameter. In the previous version, it was only possible to do that in rapid.

Driven point

- Possibility to drive a point which is not the point of the tool insert that machines the part.



Finishing operations

- Spinning operations are supported. In this case, the tool must be defined as a special milling tool.

- Trochoidal groove machining is supported.
  http://www.youtube.com/watch?v=HRtSpY0SdBY

Methods

- Possibility to automatically break the associativity between the method and the operations created by the method.
- Possibility to create a method by selecting more than one operation.
- Edition of an interactive method by using the same values which were input when the process has been run the first time.
- Comments and PP words of operations created by methods are now recalculated like the other parameters.
- Some parameters such as the machine-tool used or the material to cut have been added.
- Three new parameters have been added to manage holes in methods (available in **Edition | Options**). These parameters allow the hole intersections in method formulas to be managed.
- A new parameter allows the creation of one centering operation (and only one) even if more than one cylinder are drilled and this on all the method operations.
- A new operation called **form process execution** allows the call of another form method.
- Method loading time has been notably improved.
- A new lock option maintains the operations consecutive. This option is not available for form hole methods.
- New way to define local variables inside a method.

Tools

- Management of the length extension links.

- Management of the clamping system for the minimal tool length calculation.

Operation manager

- In the **Operations List** tab, it is now possible to see and edit (by double-clicking) the link movements and the tool change movements.
- The name of the ISO file can be seen and edited in the **Operations List** tab of the operation manager.

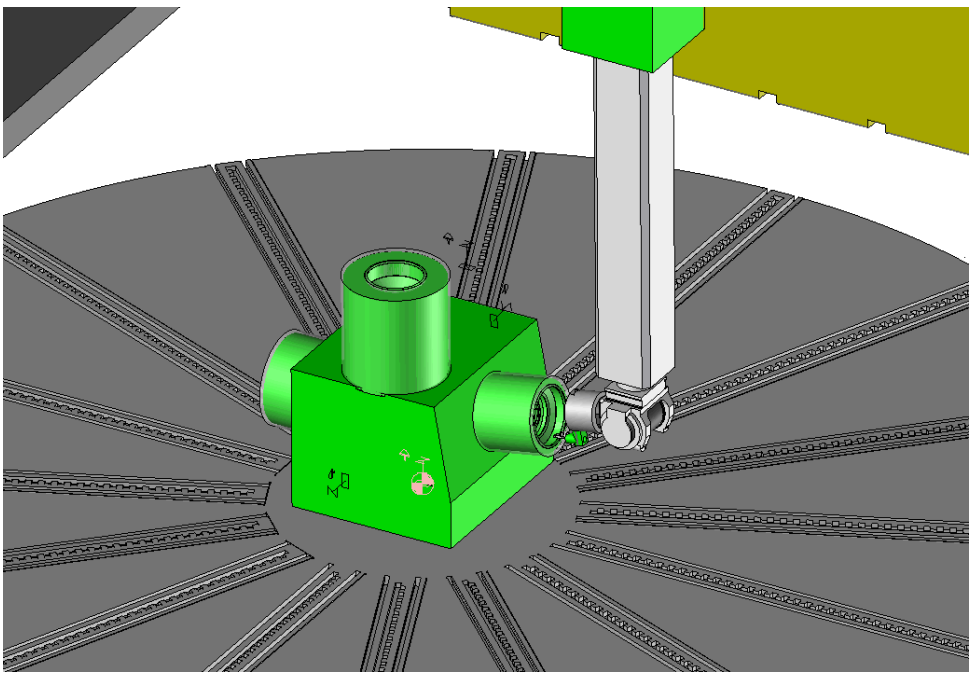
Verify

- Integration of MachineWorks 7.1 (verification of 4&5-axis operations in turbo mode).

Miscellaneous

Simulation

- The simulation of Andrea head has been improved.



- A parameter allows you to run or not the simulation after each operation creation.

Symmetry management

- A new parameter configures the way used to calculate symmetrical tool paths in contouring. In some cases, it is useful to get the exact symmetry of the original tool path. In these cases, the approach of the symmetrical operation is identical to the retract of the original operation.

User interface

- The mouse scroll is inhibited in the combo-boxes of TopSolid'Cam boxes to avoid changes of settings when the user makes a graphical zoom.

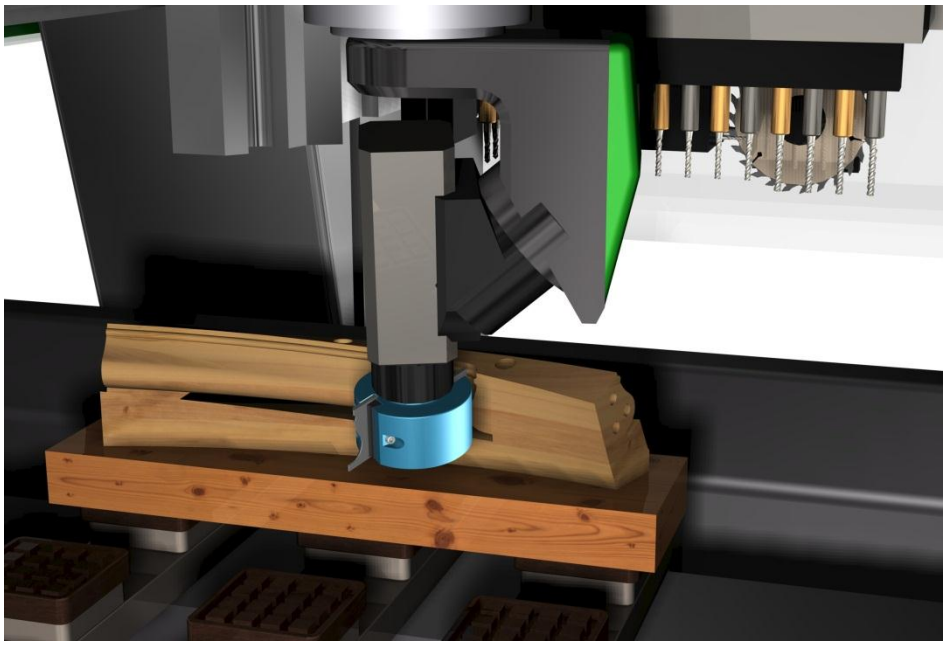
Default values

- In **Edition** | **Change default values** | **Base** | **Configuration** | **Parameters**, 9 integer, angle, length and string user parameters have been added. The goal is to customize machining methods easily. These parameters are usable as key words in method editing.
- *.CadParameters.Name* allows you to use Cad Parameters in default values of the Cam application. *Name* must be replaced by the name of the parameter.

Saving

- A new option allows the purge of intermediate stocks in order to reduce the size of the files when the user saves them.

TopSolid'WoodCam 2013: What's New



User interface

Update of menus

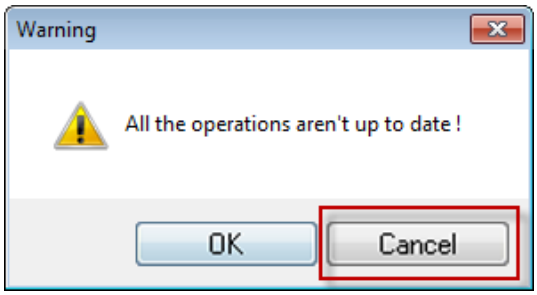
To provide continuous improvement of the user interface, the menus have been reorganized and updated.

Configuration of TopSolid'WoodCam

The **Help | Configuration** function has been enhanced to show more information such as the tool database used, the default value file, etc.

All operations are not up to date

When operations are not up-to-date and we use a function which may be affected by this state, the warning message has now a **Cancel** button to stop the function.



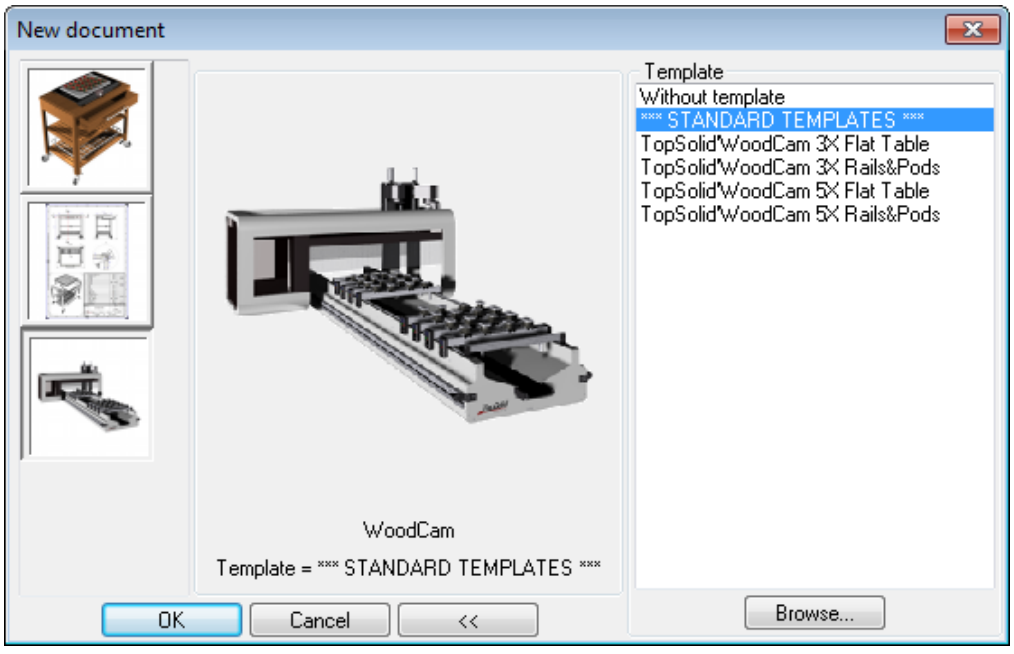
Various translations

Translations of some functions have been revised.

Machine templates

New standard machine templates

The standard machines have been replaced by generic machines illustrating the various possibilities of TopSolid'WoodCam.

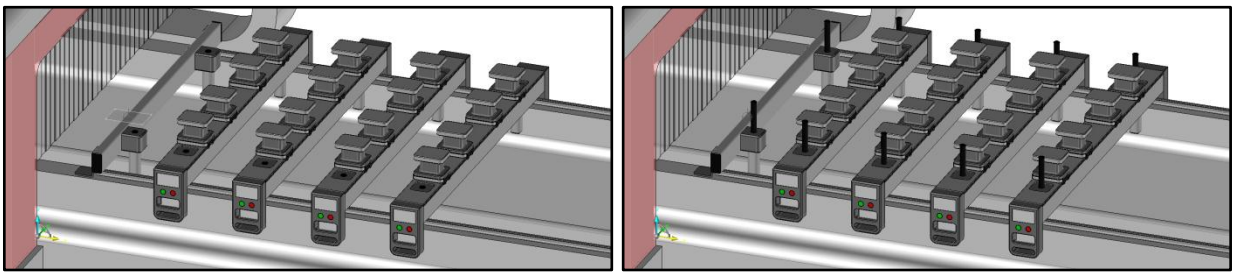


Retractable stops

Stops can be defined to be retractable. A Boolean parameter can be created and used in the machining document to indicate that stops are out or not. This parameter will be automatically modified by TopSolid'WoodCam when positioning parts.

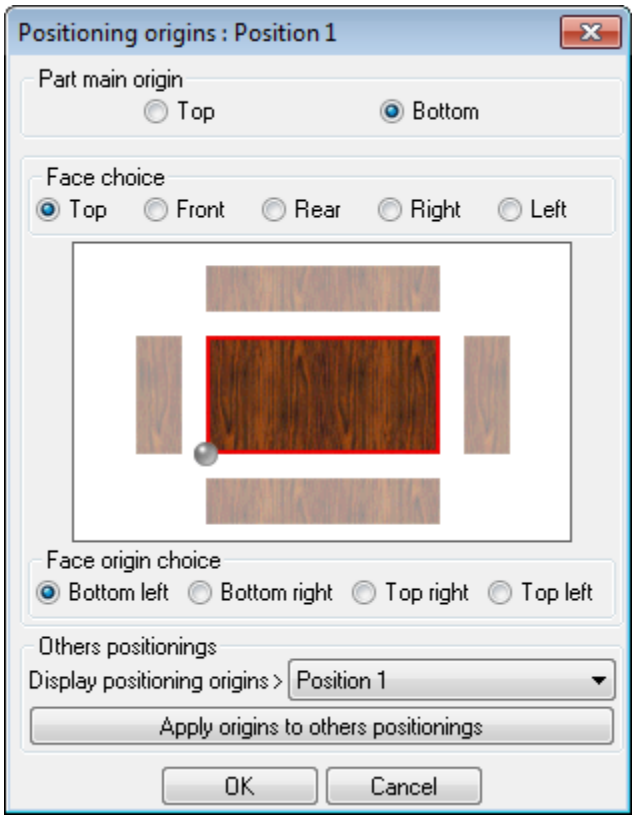
The parameter must be named *stops_activated* and its values can be:

- **0**: stops in;
- **1**: stops out.



Definition of stop origins

The window for defining origins has been revised to simplify its use.



Rotating drilling ramps

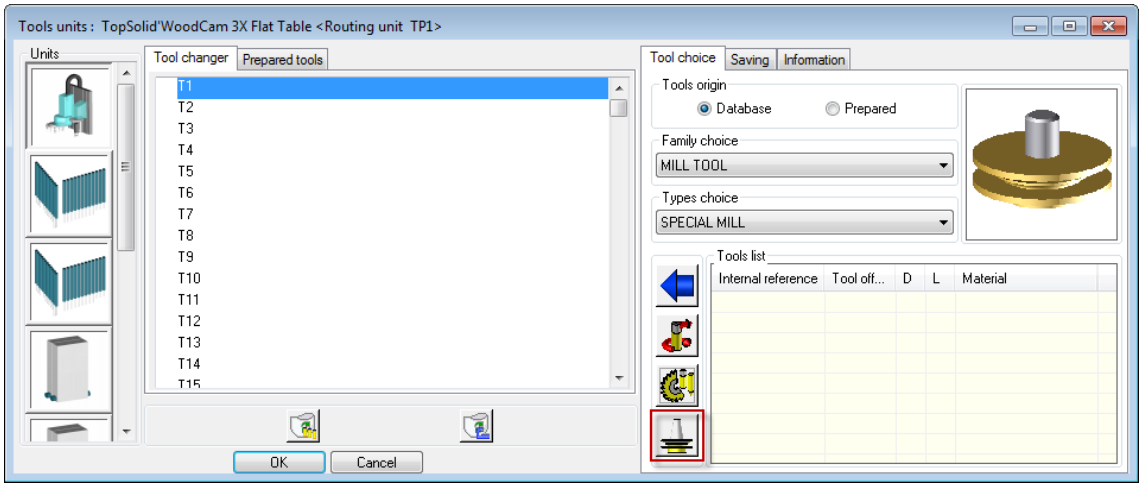
Drilling ramps can be defined with a C axis and automatically recognized for operation analysis.

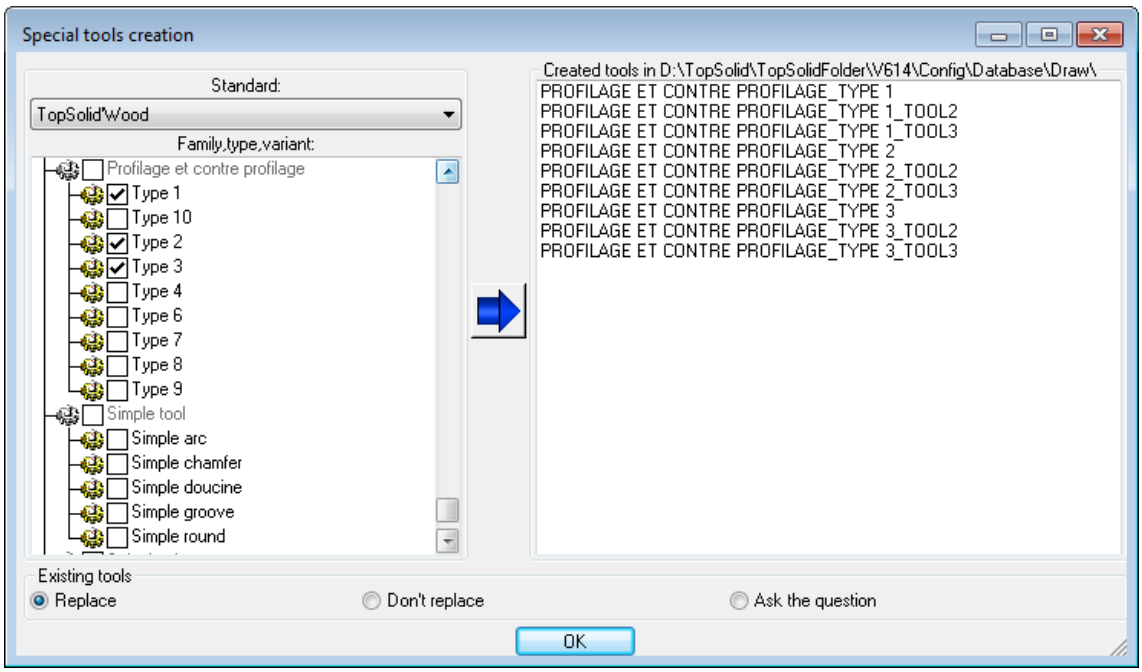
Tool management

Creation of special mills

The special mill creation can be done directly from the CAD tool library of **TopSolid'Wood**.
In the tool magazine, use the **Automatic moulding tool creation** button, and then select the tools you want to create from the list.

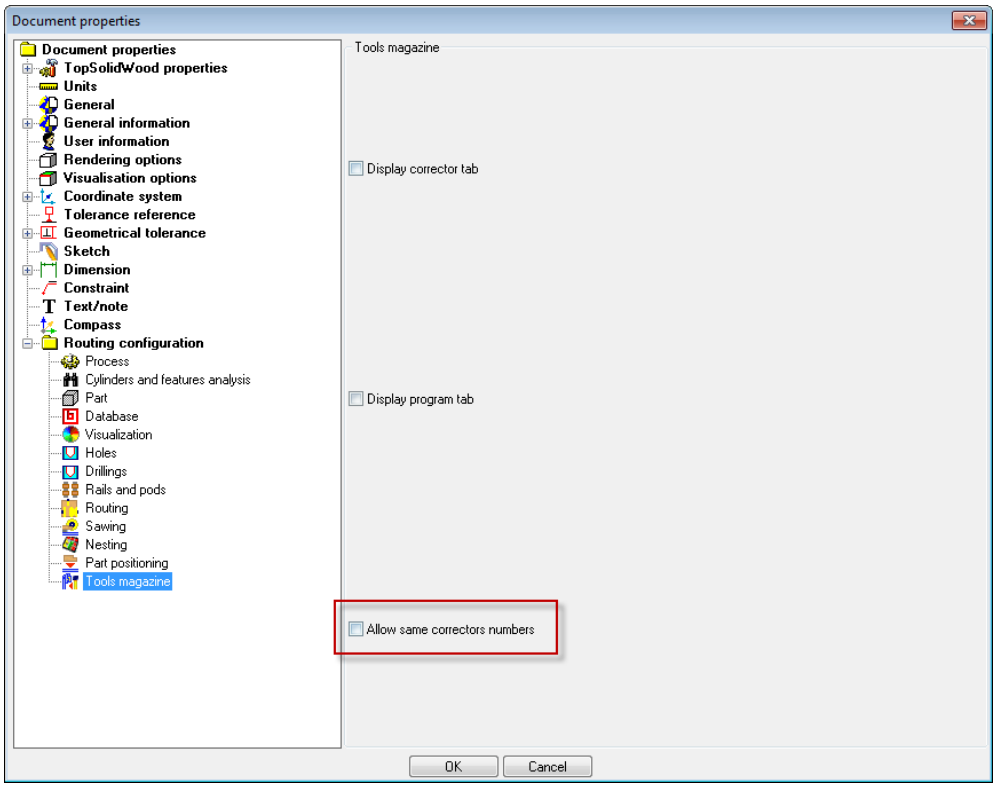
All tools are created in **Normal** mode.





Corrector number

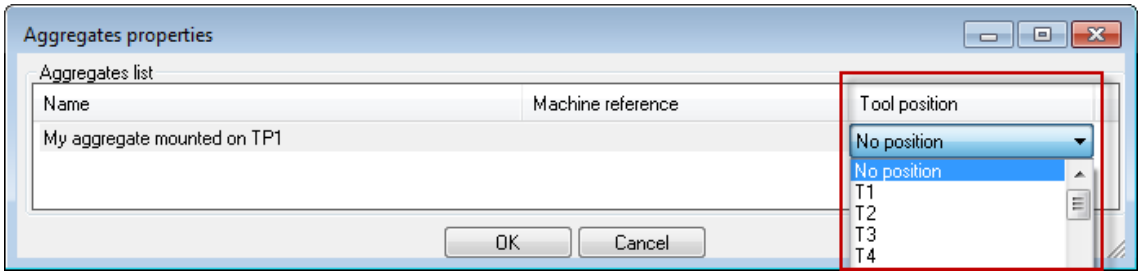
You can allow the definition of corrector number to several tools in your magazine.



Aggregates

Tool numbers of aggregates

Tool numbers of aggregates used on the main spindle can be modified after their definition.
This modification can be done using the **Equipments | Aggregates | Aggregates properties** command.



Approach and retraction macros for aggregates

Link movements for approach and retraction are now automatically defined on the aggregate, when defining it, according to its type.

Part positioning

Part positioning

To ensure positioning and repositioning of parts in different cases, the positioning functions have been improved. The enclosing shape of the part is used to avoid positioning problems depending on the part complexity.

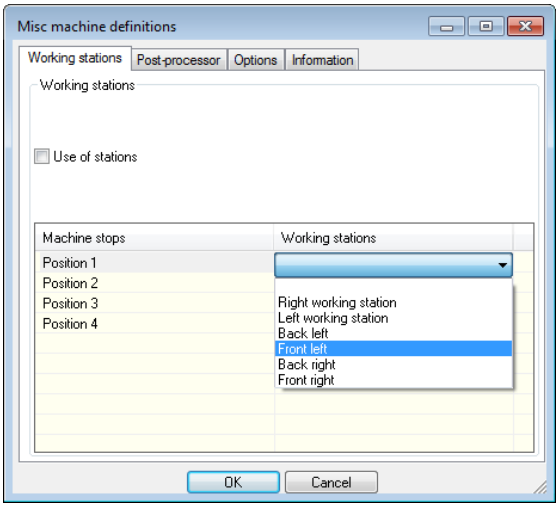
Automatic WCS creation has been improved as well.

Choosing the stop to use

If you defined the stops to use with the default working stations of TopSolid previous versions, you have to redefine these options using the new function.

In TopSolid'Wood, you can define working stations in addition to **Left working station** and **Right working station**. To create the link with TopSolid'WoodCam, you have to define the stop to be used for each working station using the **Misc. | Machines | Misc. modifications** command, and then the **Working stations** tab.

For multi-machining, the definition of working stations is important because they are used for part positioning.



Repositioning of "anyone" set

It is now possible to reposition an "anyone" set without switching back to one origin per part. The **On the stock** or **Global** mode is kept.

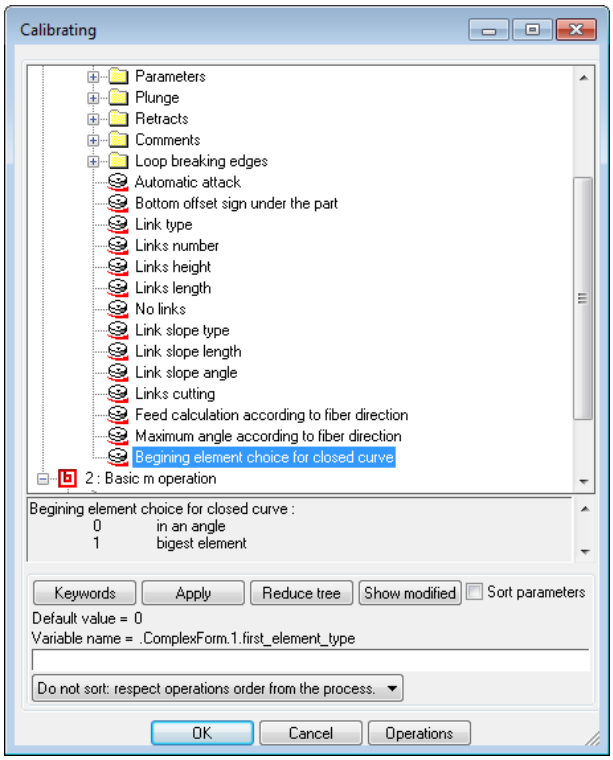
Operations

Calibrating

Starting point of calibrating operation

Calibrating the different types of parts (single, mounted set, anyone set and nesting) can now be customized for each of them.
In process associations, a different association can be done for each type of part.
A new parameter **Beginning element choice for closed curve** is available in calibrating and cutting processes to manage the starting point:

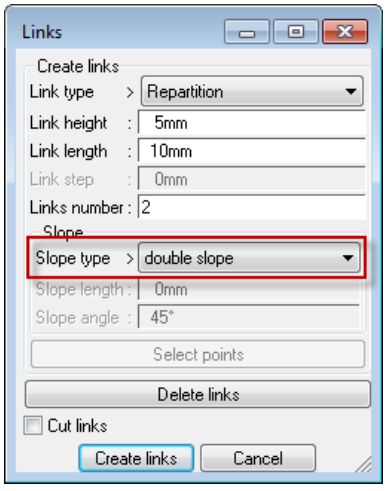
- **0**: Start in one of the angles;
- **1**: Start on the longest element;
- **2**: Start on the closest element to the center of the panel (nesting only).

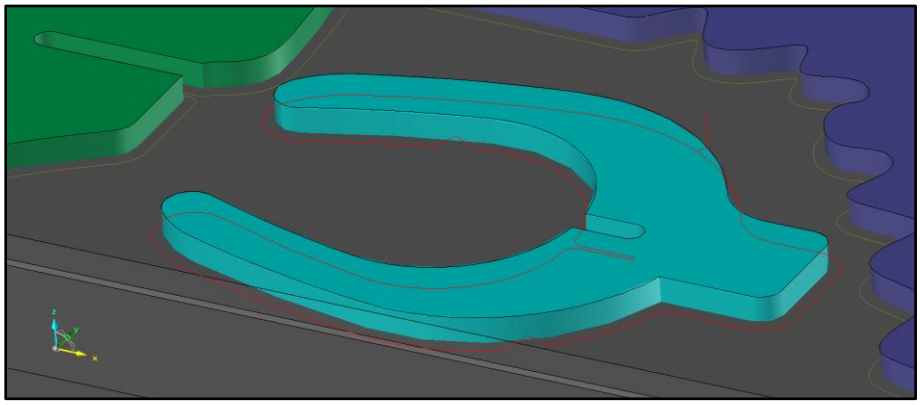


Links

Slope type

A new type of slope is available to create a slope up and down to avoid shocks to the machine.

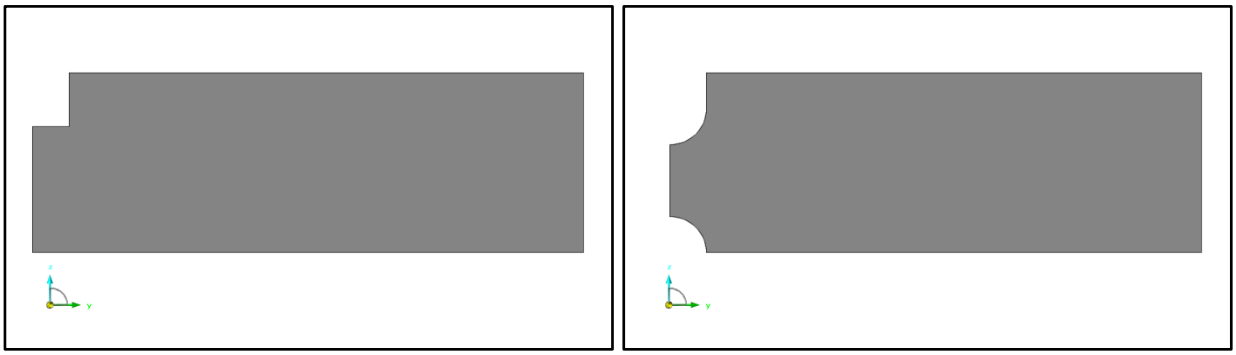




Groove, rabbet and moulding

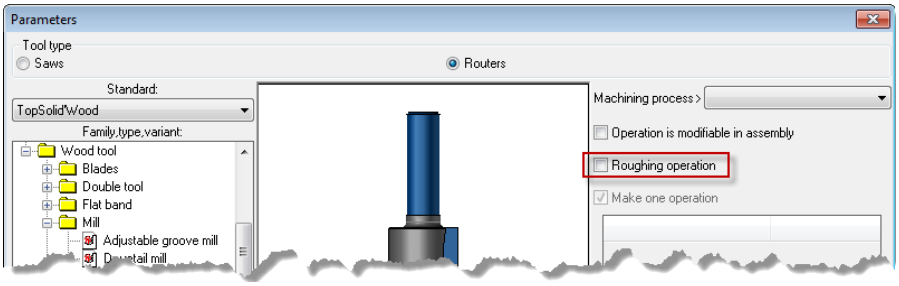
Roughing operation

When analyzing operations on the part, only operations which still modify faces of the part are analyzed and machined.
In some cases, operations are needed to make roughing operations on the part before the final operations.
In the following example, the first operation is a rabbet which removes material before the final moulding operation. The rabbet operation is totally overlapped by the moulding.

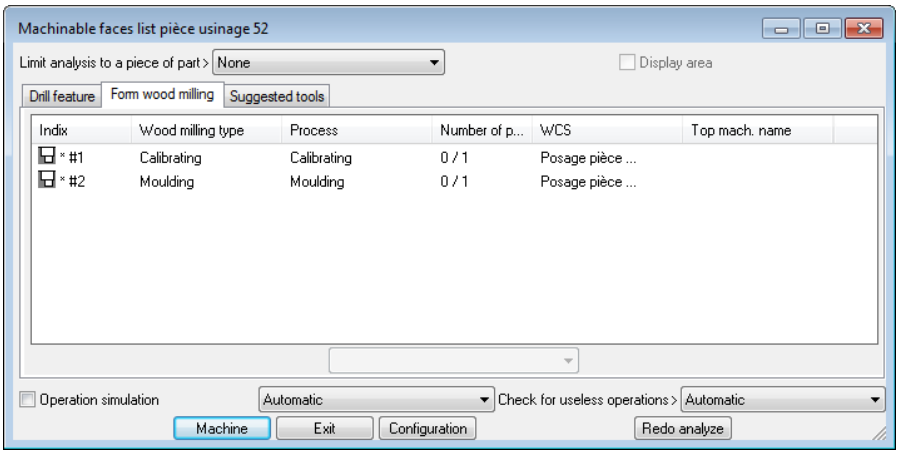


It is now possible to define a CAD operation as a roughing operation. This allows you to indicate to TopSolid'WoodCam to machine this operation, even if no more face of the operation is on the part.

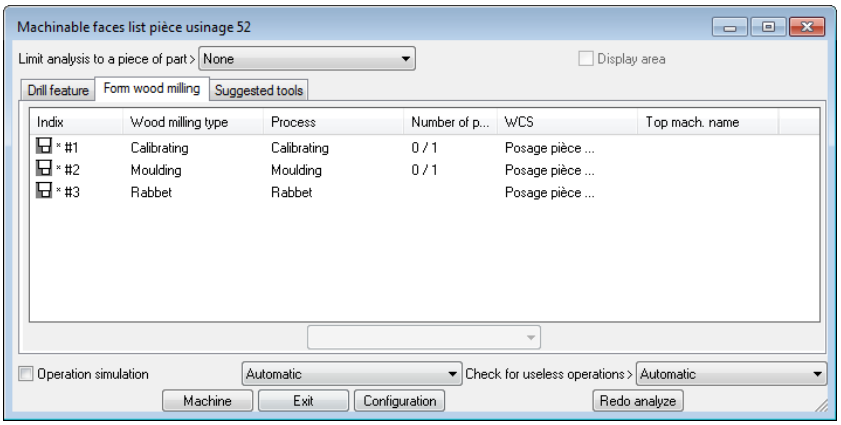
This option is available in the creation window for wood operations.



When analyzing operations, if the option is unchecked, the rabbet is not analyzed because it is hidden by the moulding.



If the option is checked, the rabbet is analyzed and machined.



TopSolid'WoodCam cannot machine propagated roughing operations. Only the first operation will be machined.

User machining

User machining on component

You can define a user machining as a tool of components. This type of operation is now machined by TopSolid'WoodCam.

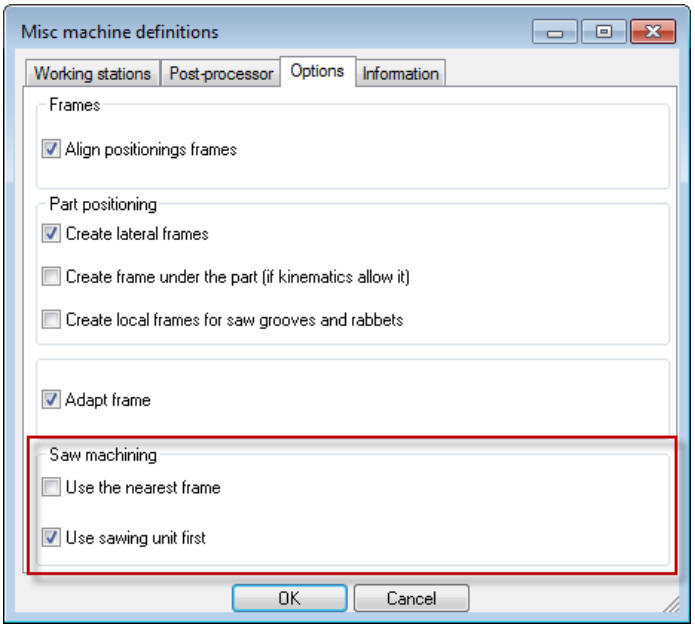
Groove and rabbet using a blade

Blade choice for wood operations

Choosing the blade for groove and rabbet machining can be set in order to select different kinds of spindles at first.
This configuration can be done using the **Misc. | Machines | Misc. modifications** function, and then the **Options** tab.

These options are:

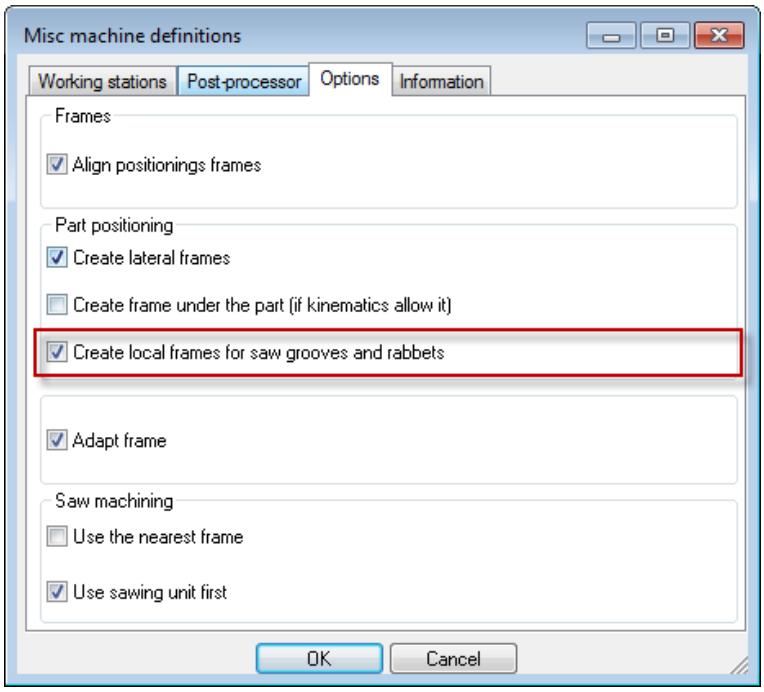
- **No selected option**: The tool orientation is done based on the CAD operation.
- **Use the nearest frame**: The nearest WCS for the operation is used first to search an available blade.
- **Use the sawing unit first**: If the operation can be done with the sawing unit, it will be used first.

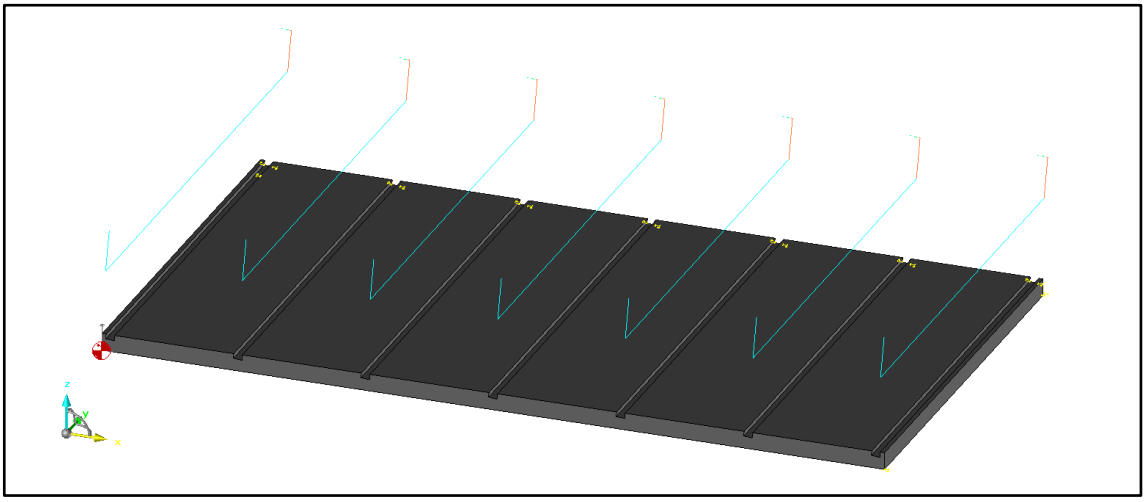


Operation with a blade on inclined plane

A new option is available to create local WCSs for wood operations performed with a blade (groove and rabbet), when the WCS is different from the main WCS, even if an existing WCS is available on the needed orientation.

This option allows you to perform inclined operations with blade on some machine types with specific security constraints.



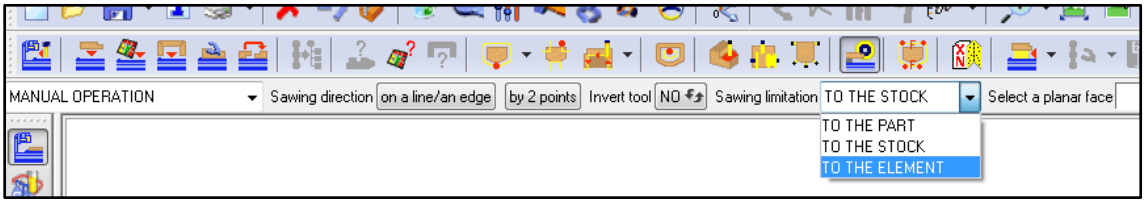


Sawing

Sawing limitation

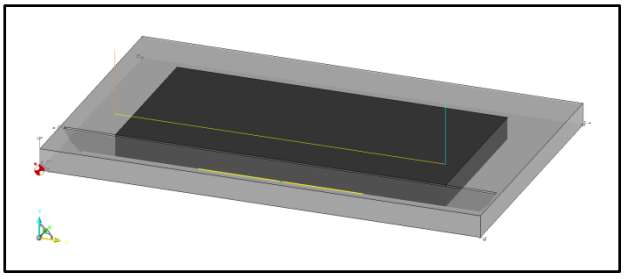
A new option for the sawing operation allows you to limit the length of the sawing according to the selected element (face, line, etc.).

This option is available for manual sawing by selecting the limitation type or by using processes.

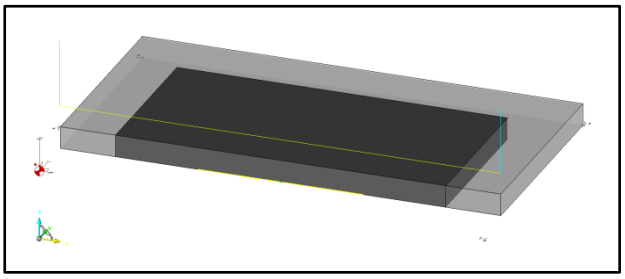


Three options are available:

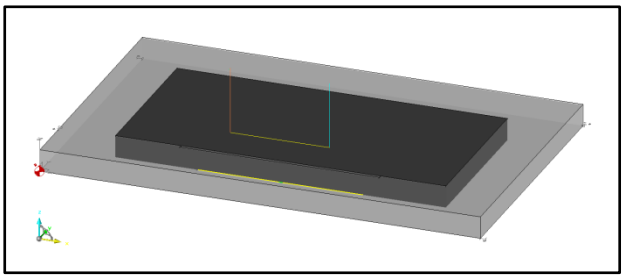
- **To the part**: The sawing path is extended to finish part limits.



- **To the stock**: The sawing path is extended to stock limits.

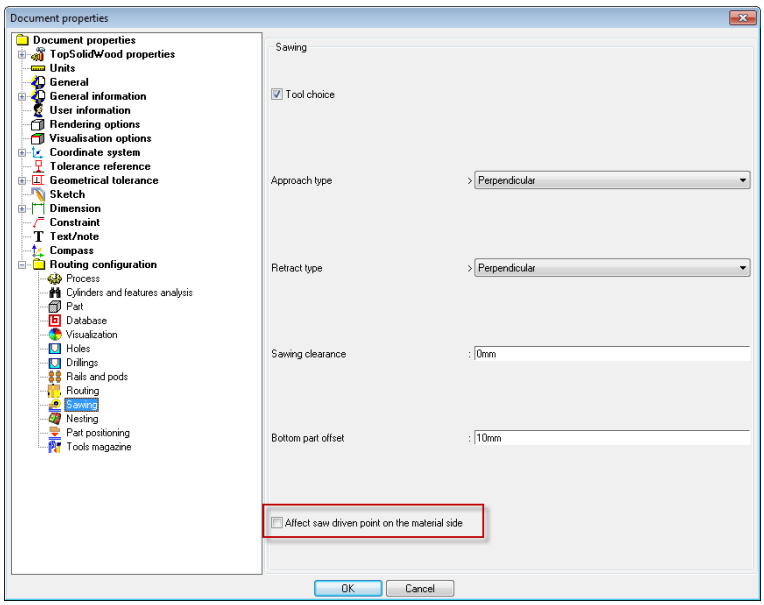


- **To the element**: The sawing path is trimmed to the dimensions of the selected element.



Driven point choice of blade according to material side

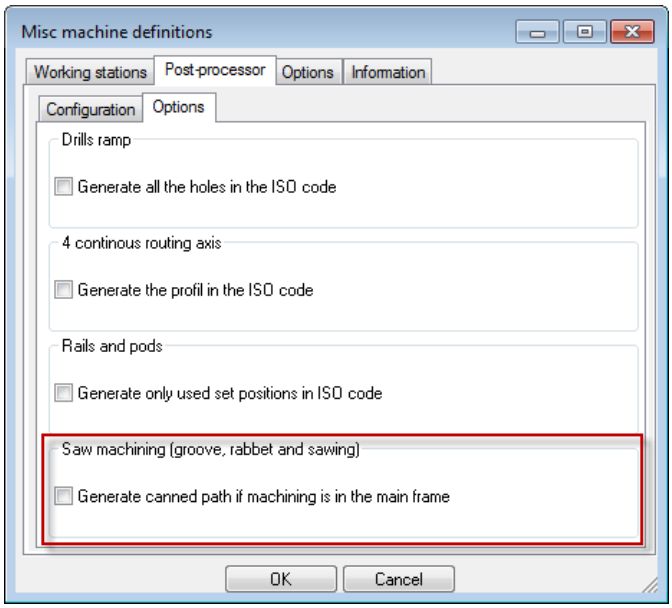
TopSolid'WoodCam can automatically select the driven point of a blade for sawing operations in order to set it on the material side.
We will have the bottom driven point when we will use the front face of the blade, and the top driven point when we will use the back face of the blade.



Operation using a blade

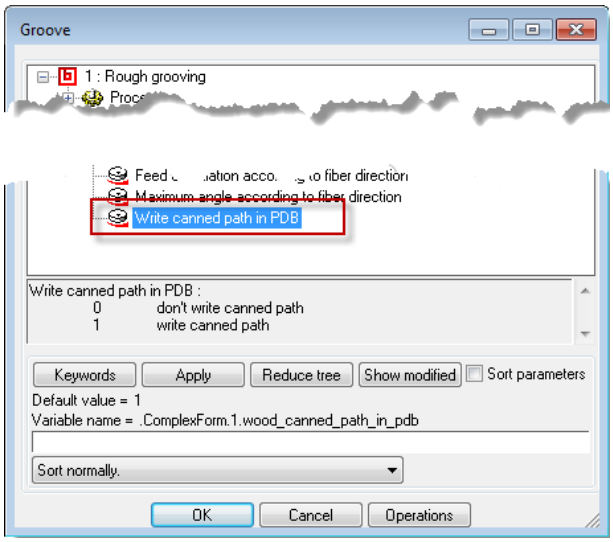
Operation using a blade in the main WCS

An option is available to generate the canned path in the PDB for wood operations using a blade in the main WCS.



Tool path

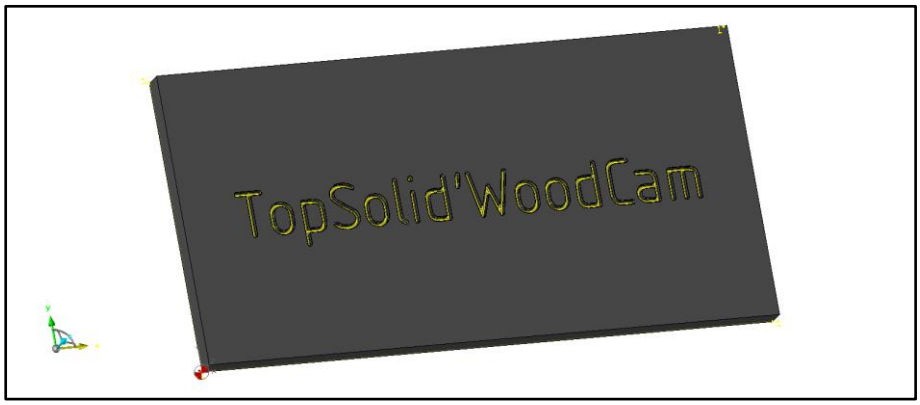
The new parameter **Wood canned path in the PDB** available in the default values and the processes allows you to generate or not the canned path in the PDB.



Pocket

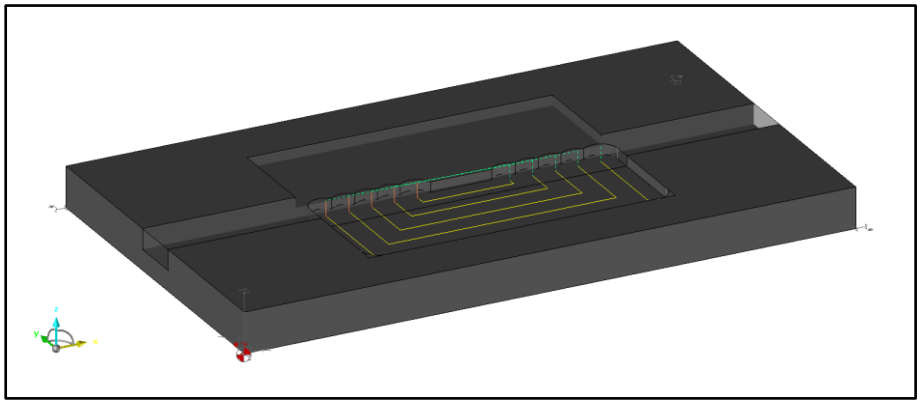
Multi-contour pockets

Automatic machining of pockets done from multiple contours (text, …) is now possible.

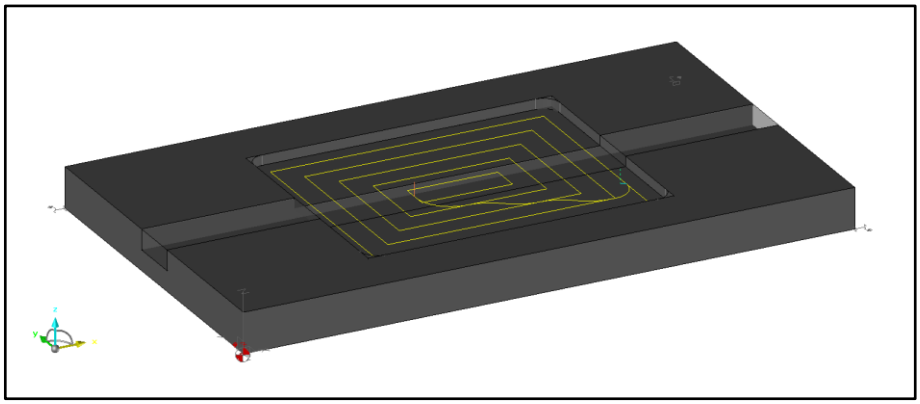


Pockets cut by another operation can also be machined as one pocket and not only as open pockets.

Result with version 6.13:



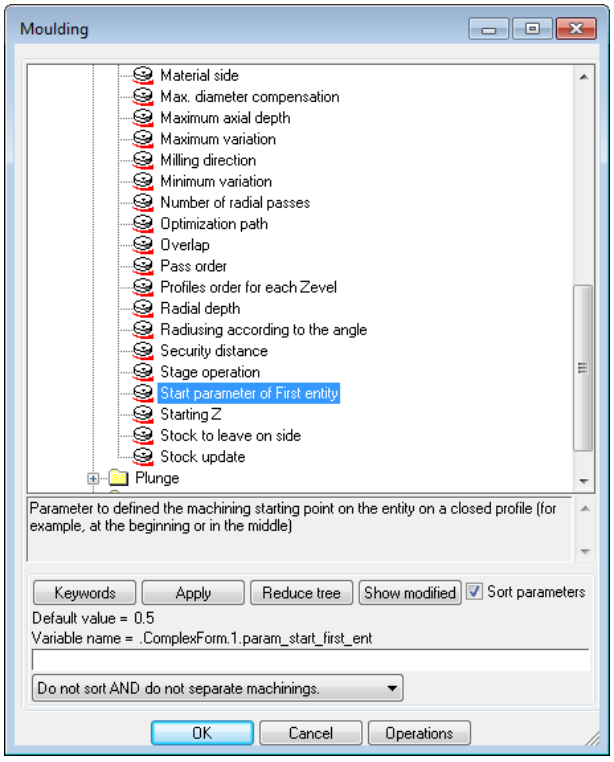
Result with version 6.14:



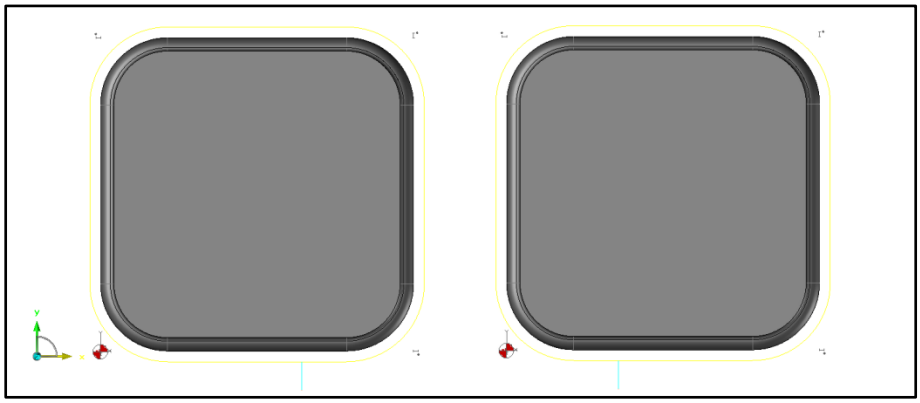
Common

Starting point on closed contour for wood operations

Until now, the starting point of wood operations was at the beginning or at the middle of the first element of the closed path.
Now you can choose the starting point by indicating a value between **0** (beginning) and **1** (end) (0 <= X < 1). The value **0.5** will be a starting point at the middle of the element.
The **Start parameter of first entity** parameter is available in the default values and the processes.

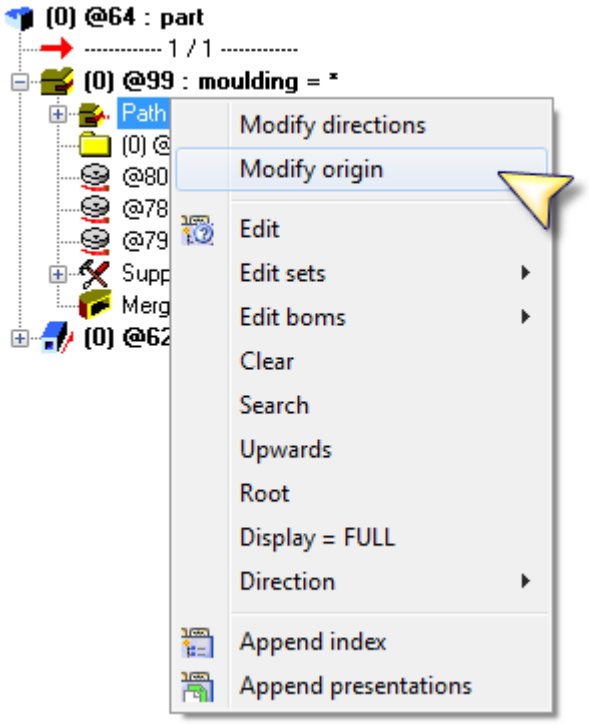


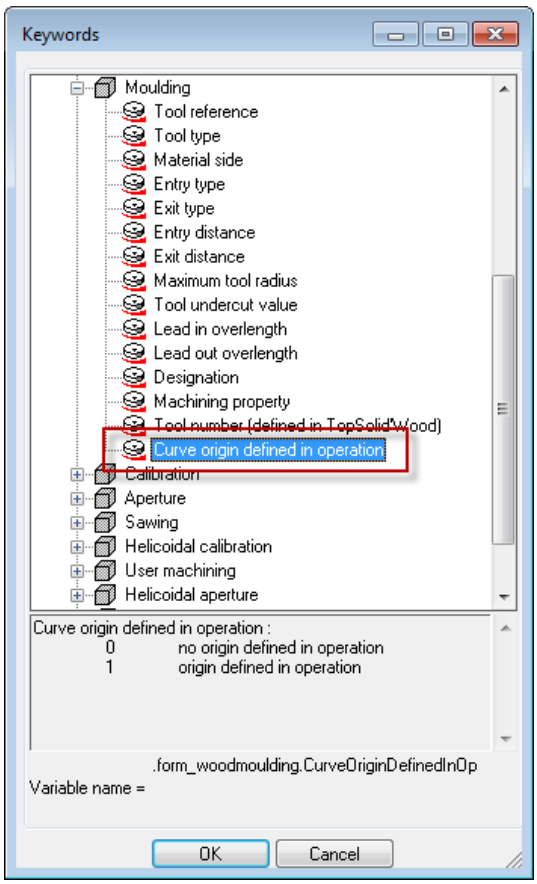
Moulding with values set at 0.25 and 0.75:



Starting point of wood operations defined in CAD

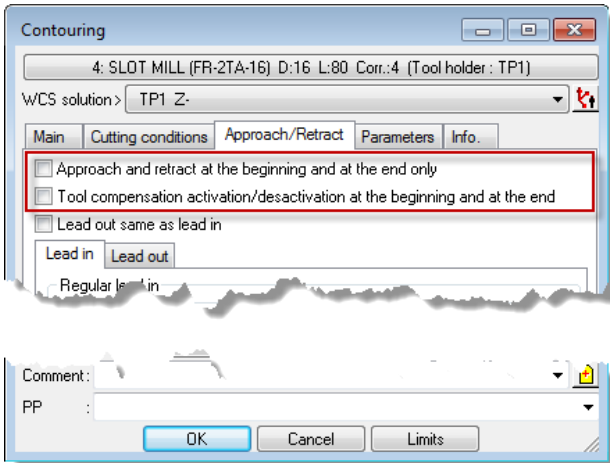
The definition of the origin point of a wood operation on a closed path will be automatically used for machining. It is possible to test in the processes whether this point is defined or not in CAD.





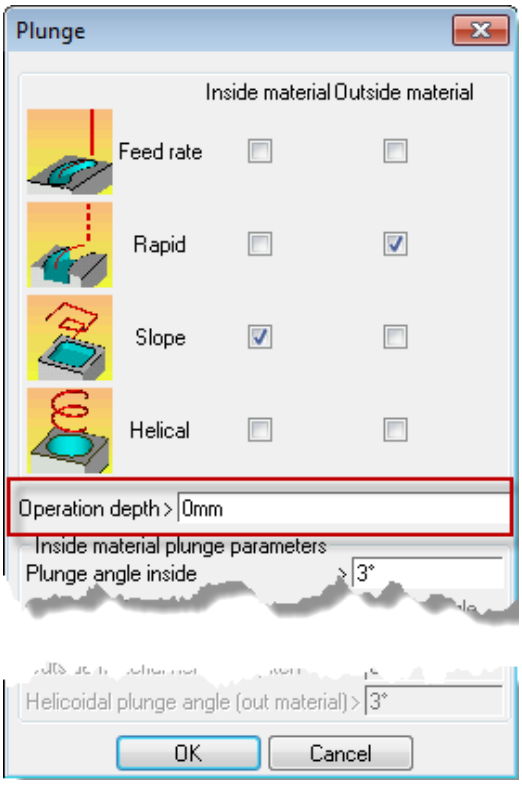
Multi-pass tool path

Approach and retraction can be done only at the beginning and at the end of the path and not at each pass. Activation and deactivation of the cutter compensation can be done in the same way.

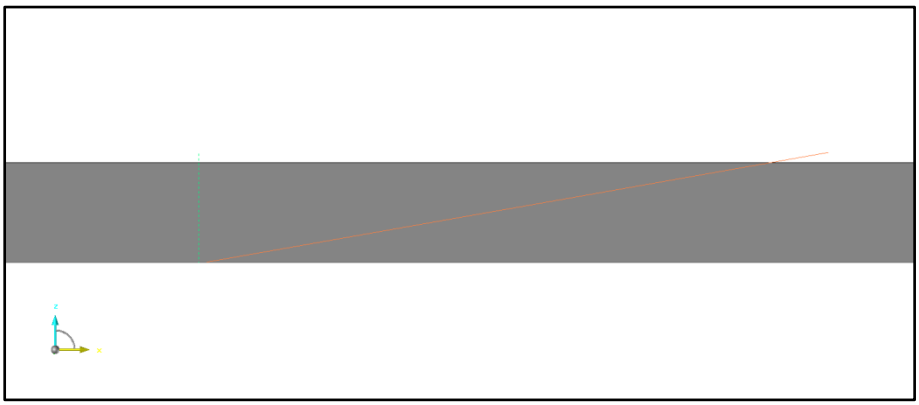


Slope plunge

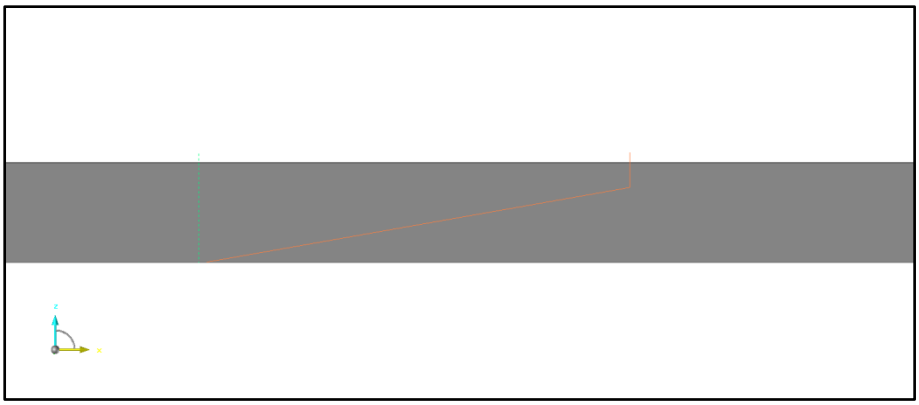
The beginning of the slope plunge is now performed from the Z level of the security distance if the operation depth value is set to 0mm.



Slope plunge with operation depth at 0mm:

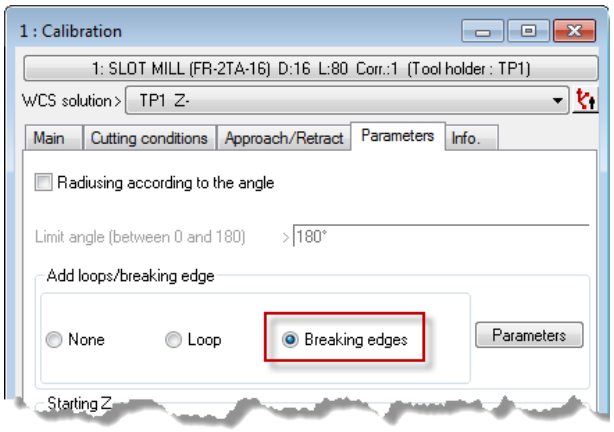


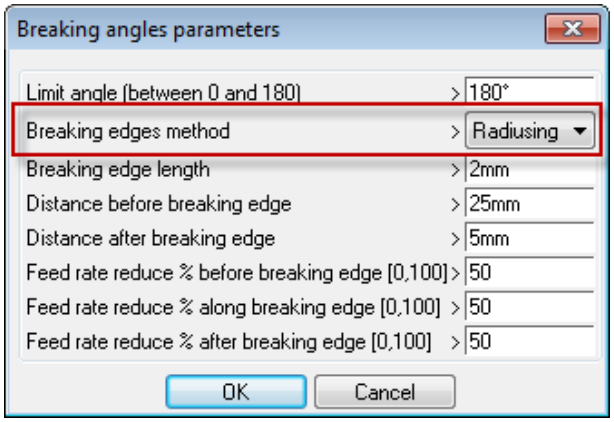
Slope plunge with operation depth at 5mm:



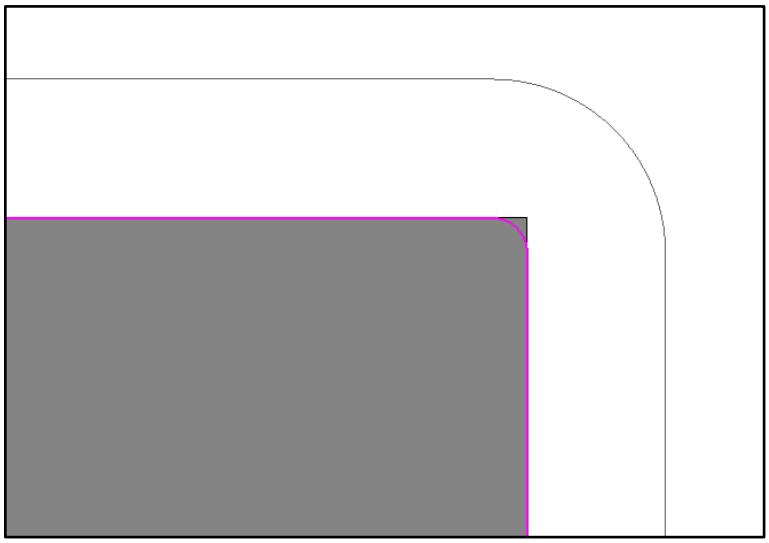
Breaking edges

A new option for contouring operations allows you to break edges with a radius.
Parameters are the same as those for breaking edges with a chamfer.





Result in purple of the breaking edges:



Movement to the clearance shape

We do not go back to the clearance shape by default anymore if we are already outside of it.

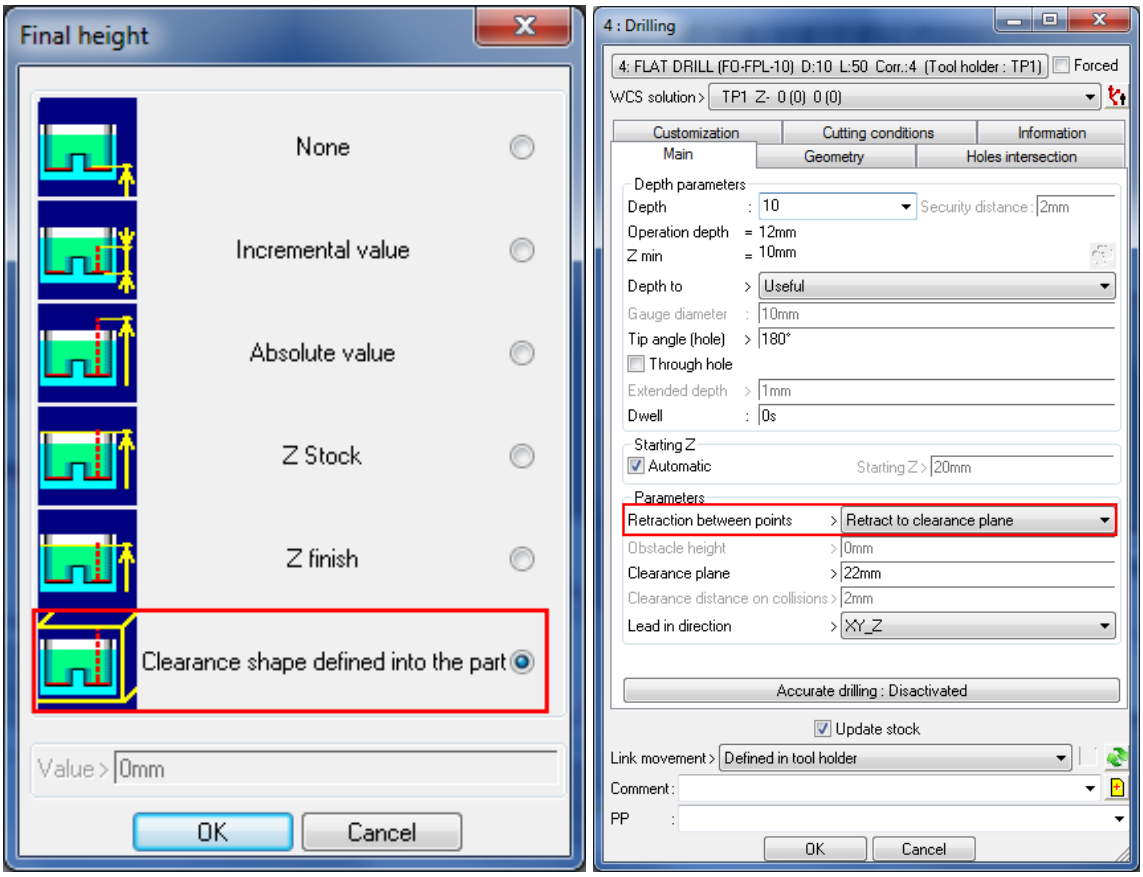
A Boolean keyword defines this behavior: **ZMI_WOOD_MACHINE_GO_ON_SECU_SHAPE**

- **0**: TopSolid'WoodCam does not move.
- **1**: TopSolid'WoodCam moves back to the clearance shape.

If the keyword is missing, the default value is set to **0**.

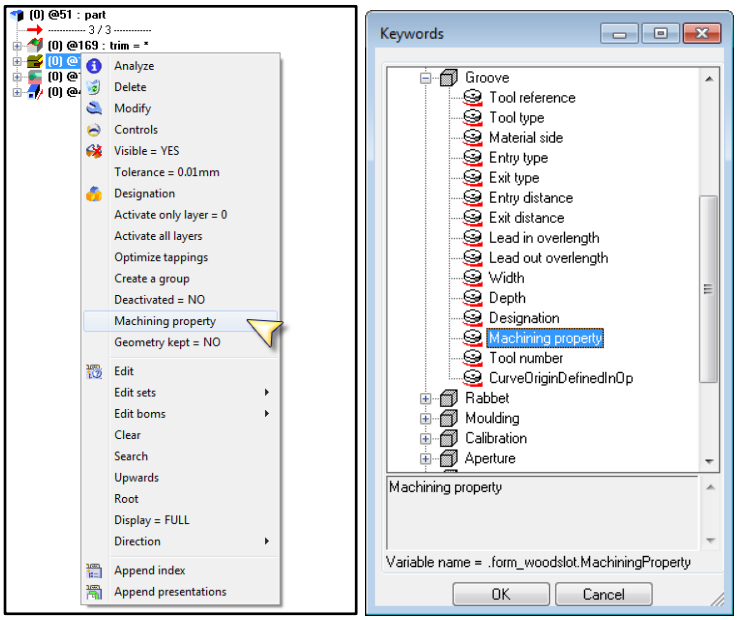
Retraction to the clearance shape

A new retraction type has been added to make a retraction to the clearance shape within the different routing and drilling operations.



Machining properties

The machining properties defined in CAD on operations can be used within processes using the keyword **Machining property** available in the different operations or directly in the PDB with the keyword **CAD_COMM**.



Retraction link movement to tool change point in 5-axis

The retraction link movement to tool change point in 5-axis (5XTool) has been improved to avoid useless movements.

Retraction link movement for spindles without tool changer

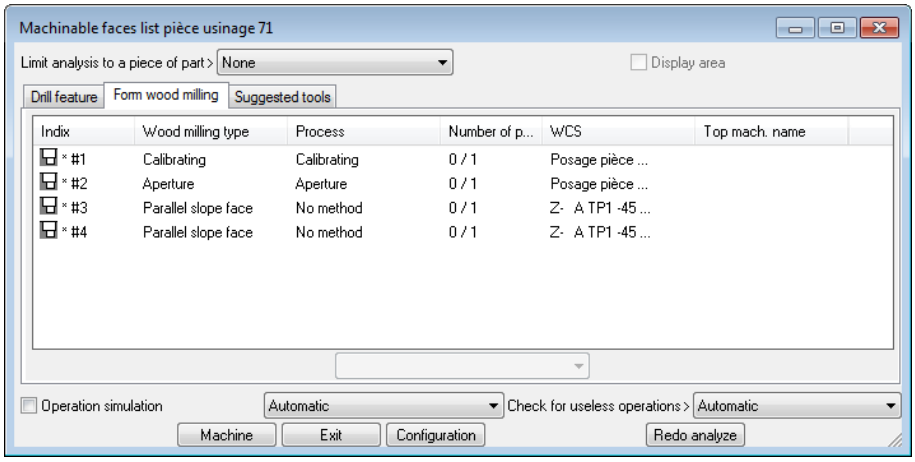
Retraction link movements are now used for spindles without tool changer when operations using them are not consecutive.

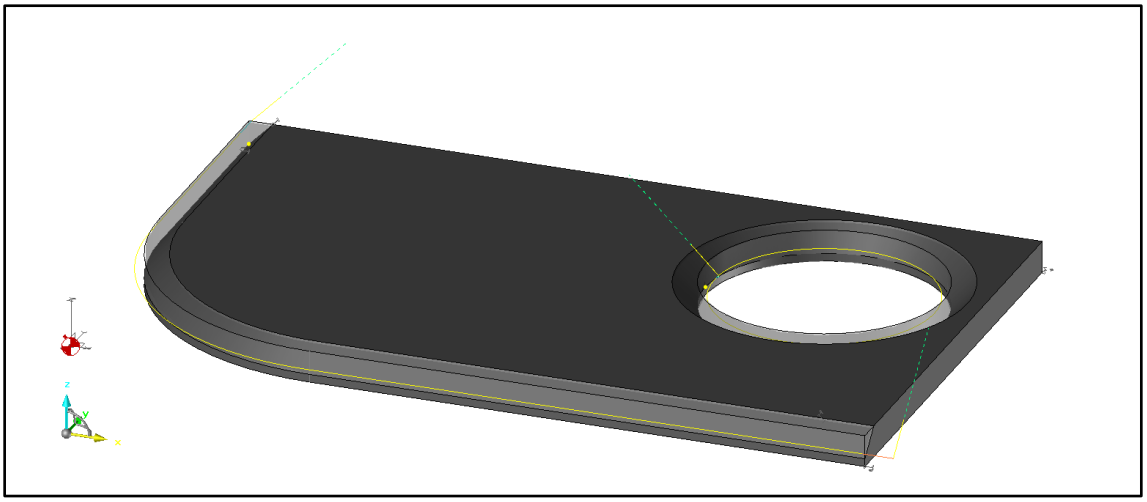
Operation analysis

5-axis continuous user machining

5-axis continuous user machining can be machined within operation analysis.

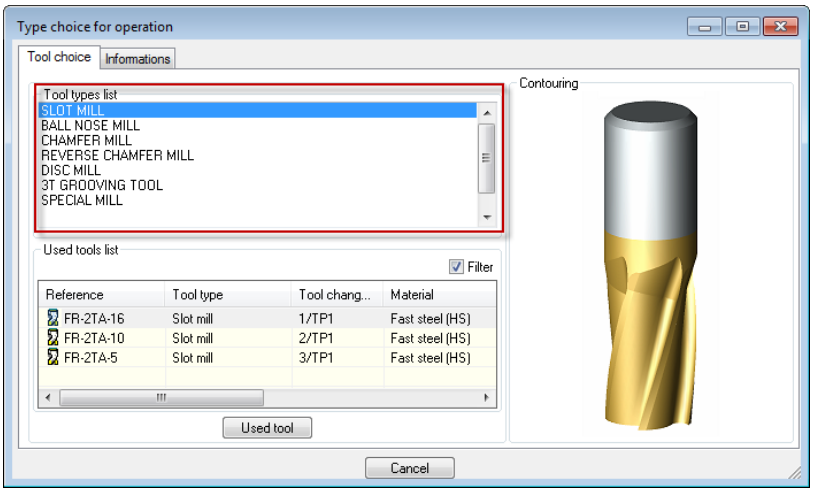
This new type of operation does not use any process; therefore the configuration of the CAD operation and the configuration of default values in CAM are very important to ensure a correct result.





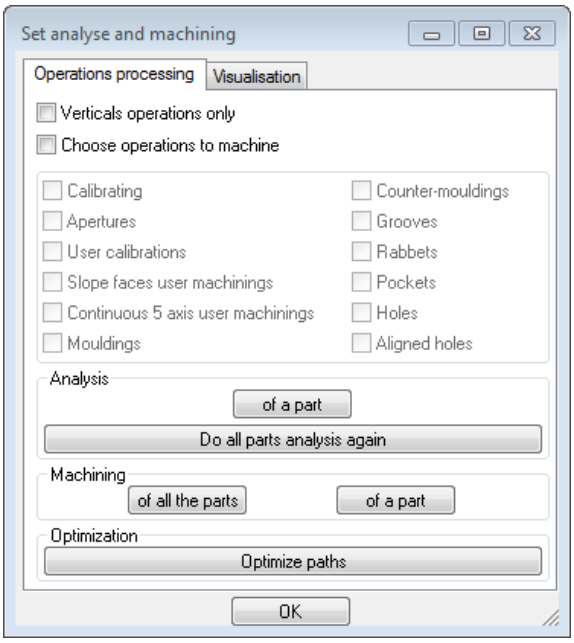
Available tool types for operations

Default available tool types for TopSolid'WoodCam have been revised and updated according to their use.



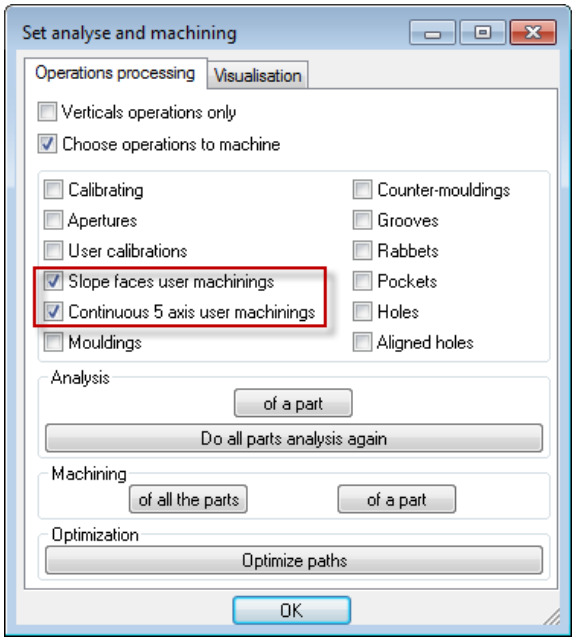
Operation analysis for mounted set

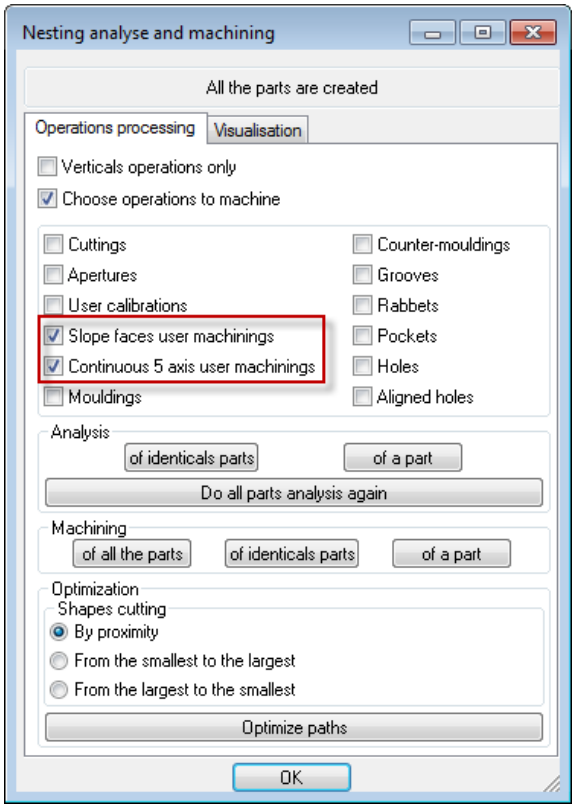
Operation analysis of sets has been harmonized so that the analysis is the same for all types of set.
You can now make an overall analysis of the whole set or perform analysis part by part as before.



Additional operation types for set and nesting machining

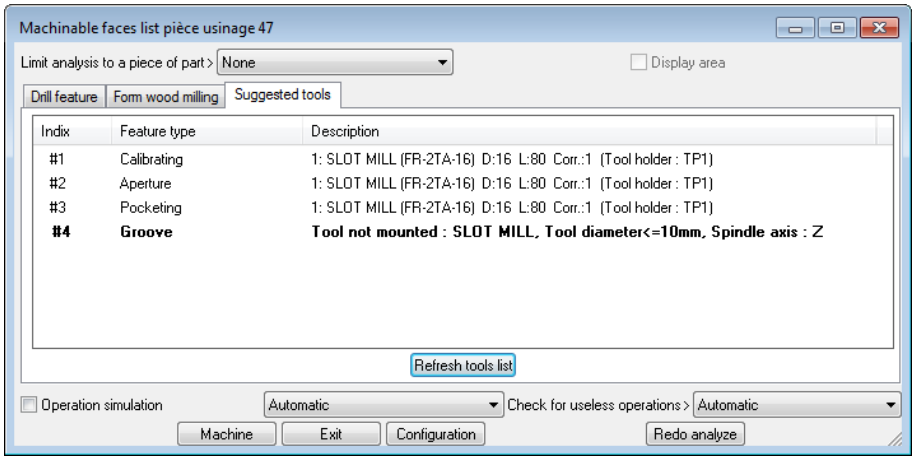
Slope faces and continuous 5-axis user machinings have been dissociated from standard user machining.





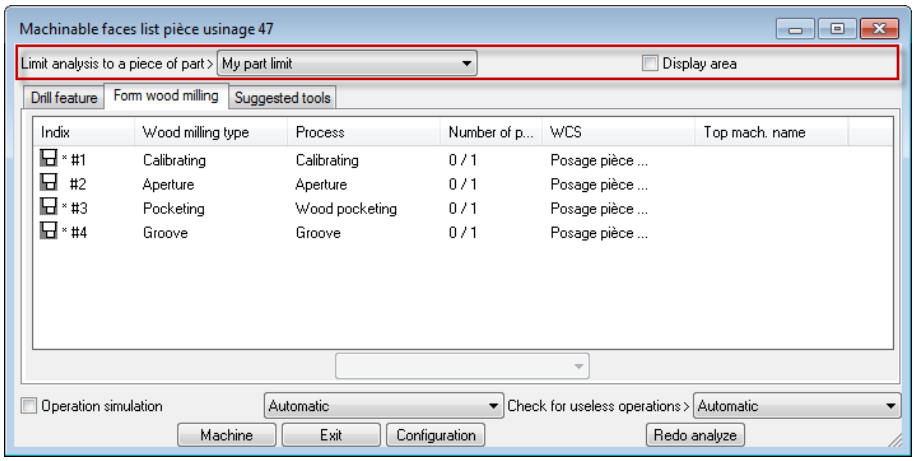
Missing tools

Missing tools are shown directly in the **Suggested tools** tab and no longer in the alpha bar.



Part limit analysis

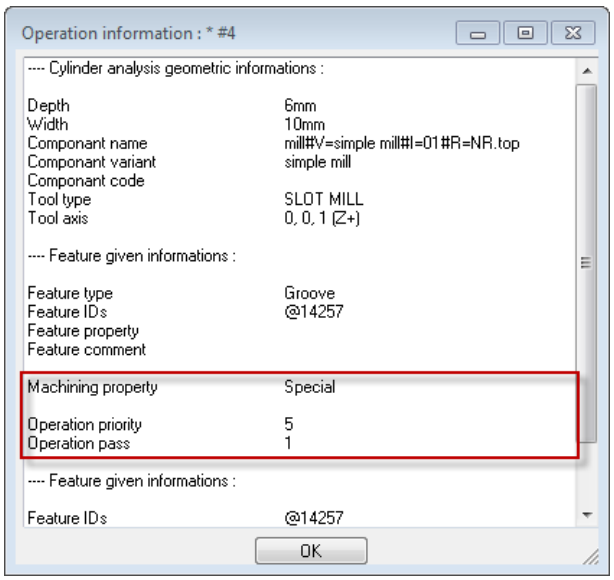
Aperture detection now uses part analysis limit. Only apertures within or intersecting the part analysis limit will be machined.
Manual aperture machining will use contours found by the operation analysis with the part analysis limit which is used.



Operation information

Additional information has been added in the operation information window. You can access the operation information window using the **Wood machining | Information** command or during the operation analysis:

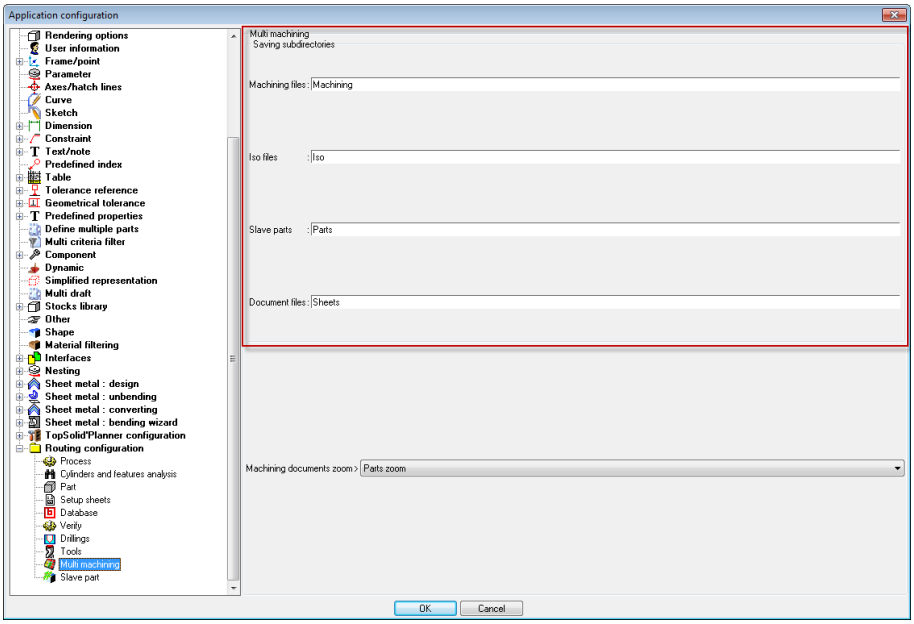
- **Machining property**;
- **Operation priority**;
- **Operation pass**.

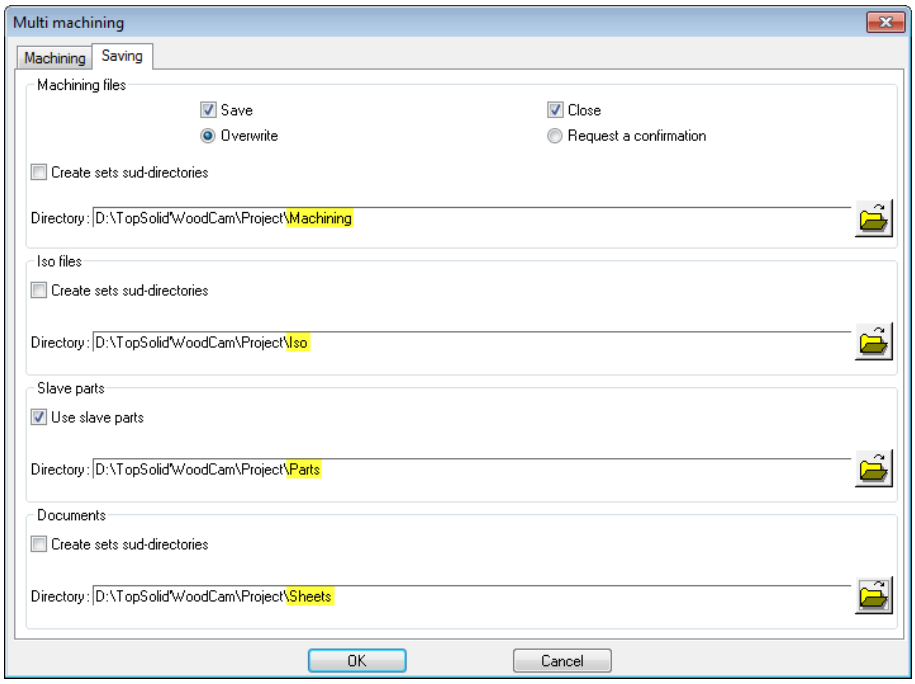


Multi-machining

Default saving sub-folders

You can define default sub-folders to save machining files, ISO files, slave parts and setup sheets when running multi-machining in **Tools | Options**, and then in **Routing configuration | Multi-machining** section.



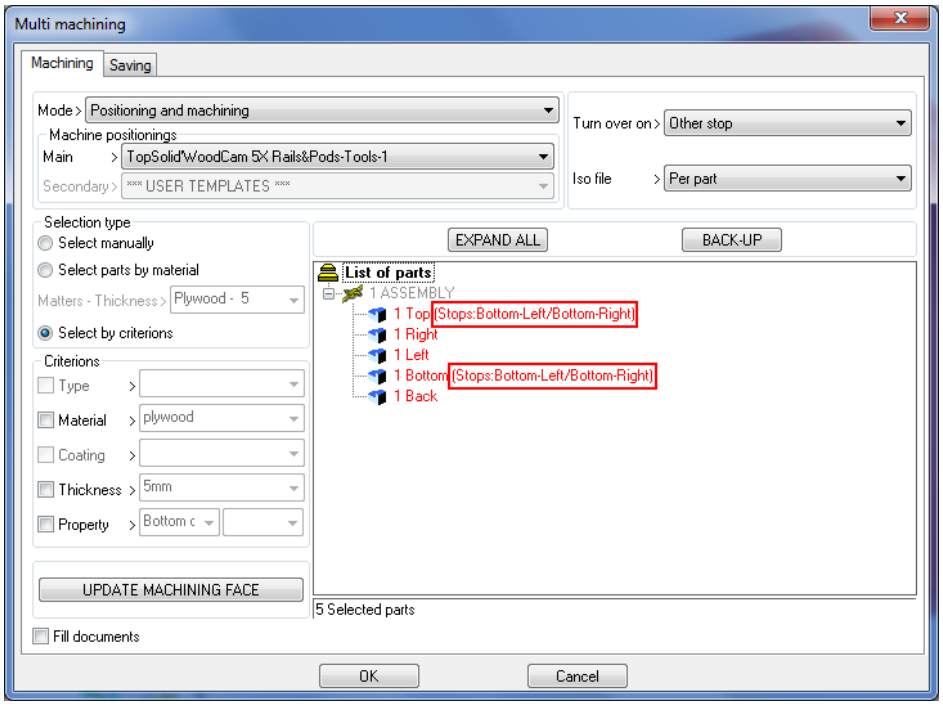


Launching options of multi-machining

BOM sorting depth and BOM filter are now stored to keep the last used options.

Modification of the stops to use for multi-machining

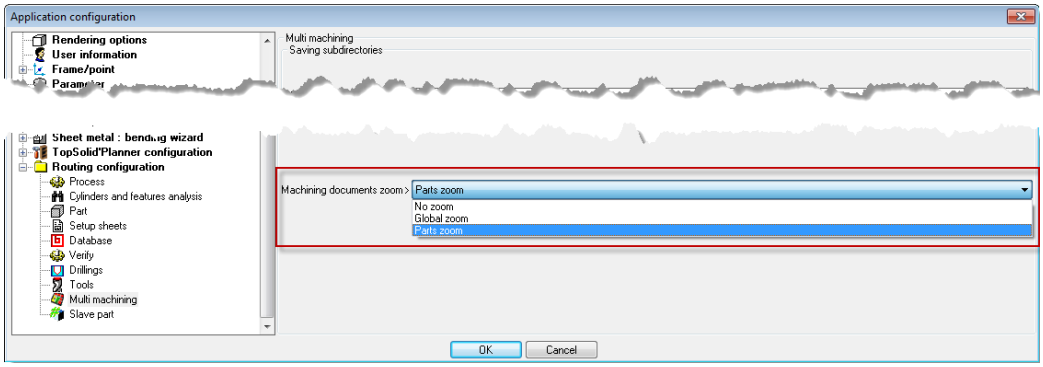
When the stops to use for positioning are modified, they are shown after the name of the part/nesting.



Type of zoom during multi-machining

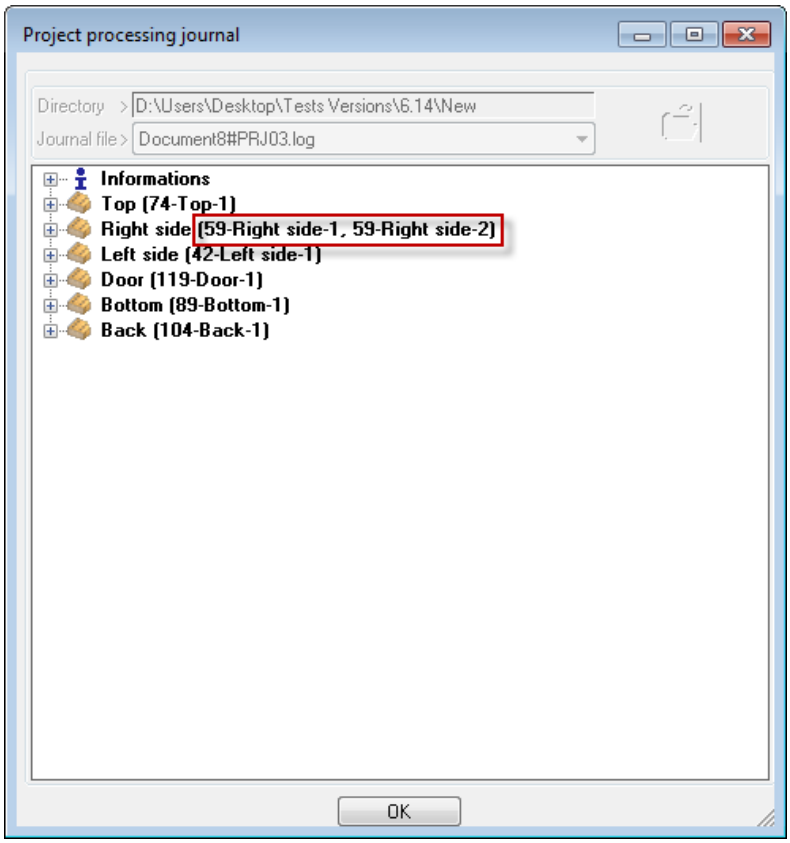
You can manage the zoom done at the end of the multi-machining:

- **No zoom**;
- **Global zoom**;
- **Parts zoom**.



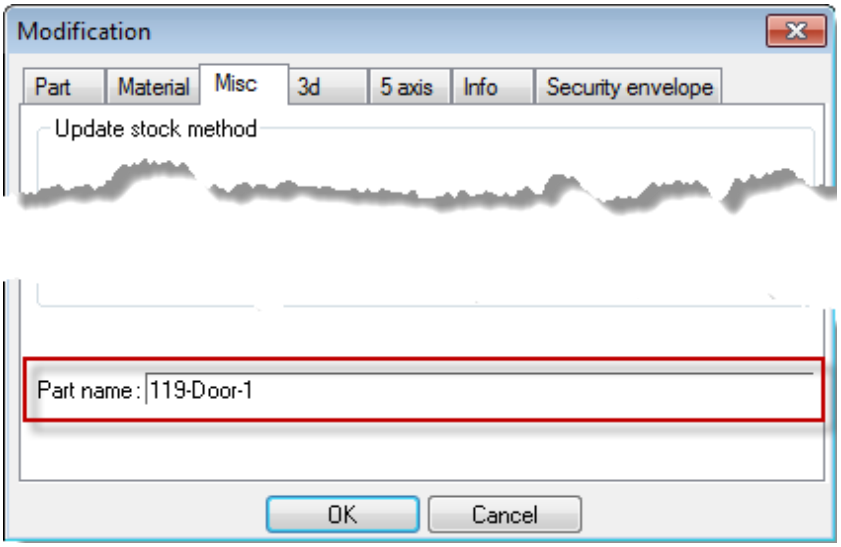
Machining name in the multi-machining journal

Names of machining files, defined by **WOO_CAM_FILE_NAME** and **WOO_CAM_FILE_NAME2**, are shown after the designation of the path in the project processing journal.



Part name

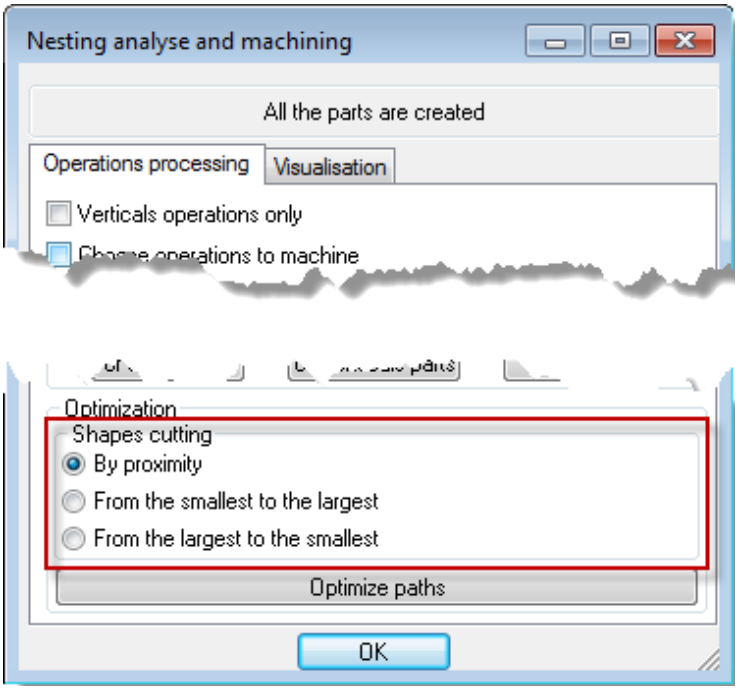
The name of the machined part is defined by **WOO_CAM_FILE_NAME** or **WOO_CAM_FILE_NAME2** according to the positioning side of the part.



Nesting machining

Optimization of nesting cutting

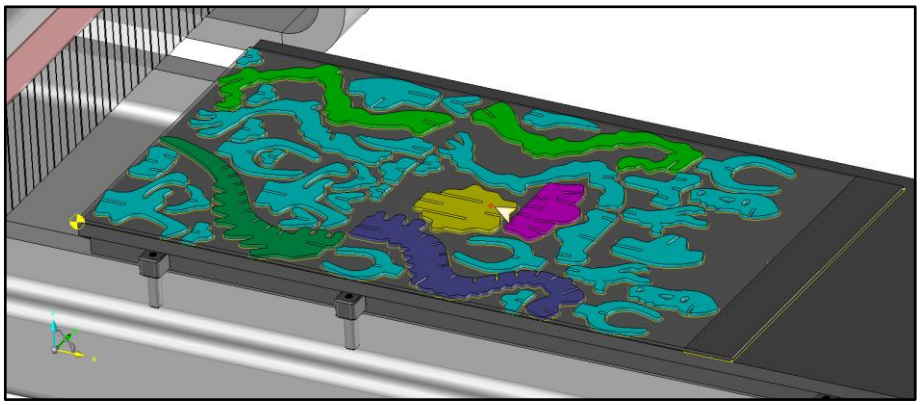
Cutting paths of nesting can be sorted from the smallest to the largest part and vice-versa.
Optimization by proximity is still available.



Nesting cutting

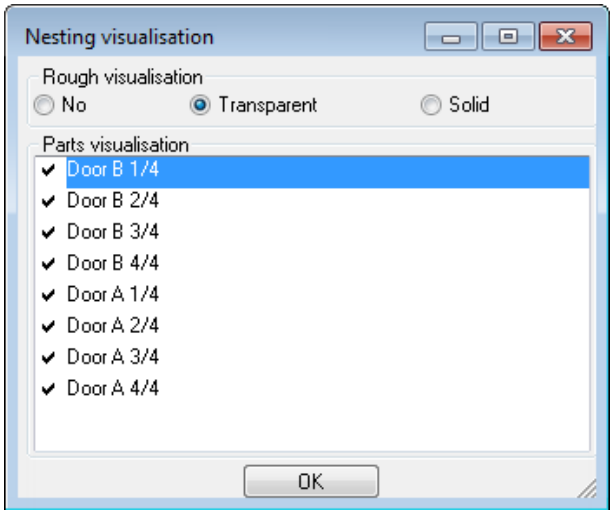
The starting point of the cutting operation is set to be as close as possible to the center of the panel to keep the outside parts held to the panel as long as possible.
This new mode is automatic but it can be deactivated in the default values or the processes.

This mode is only available if the nesting is positioned using the **Create all the parts** option.



Part visualization of nesting

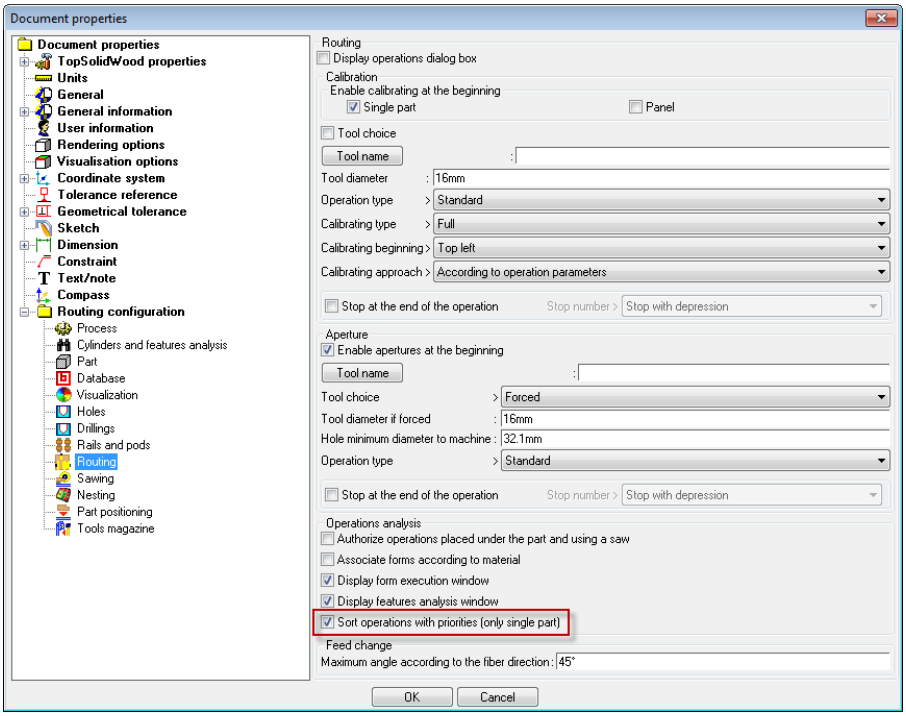
When the nesting positioning is done using the **Create all the parts** option, the visualization of parts with the **Part | Visualize** function lists all the parts and not only one part per family.



Operation management

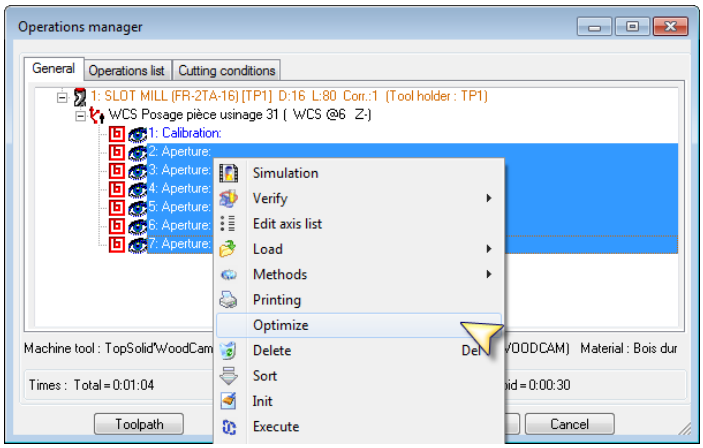
Sort operation by priorities

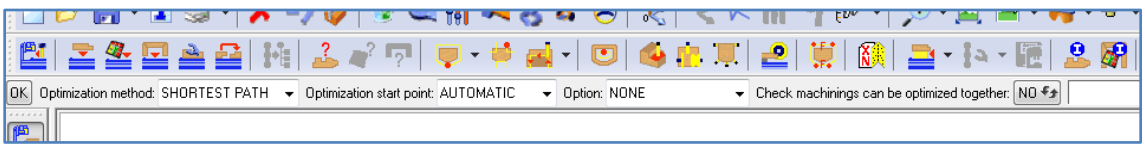
Sorting by priorities can be automatically done after executing operation from the operation analysis.

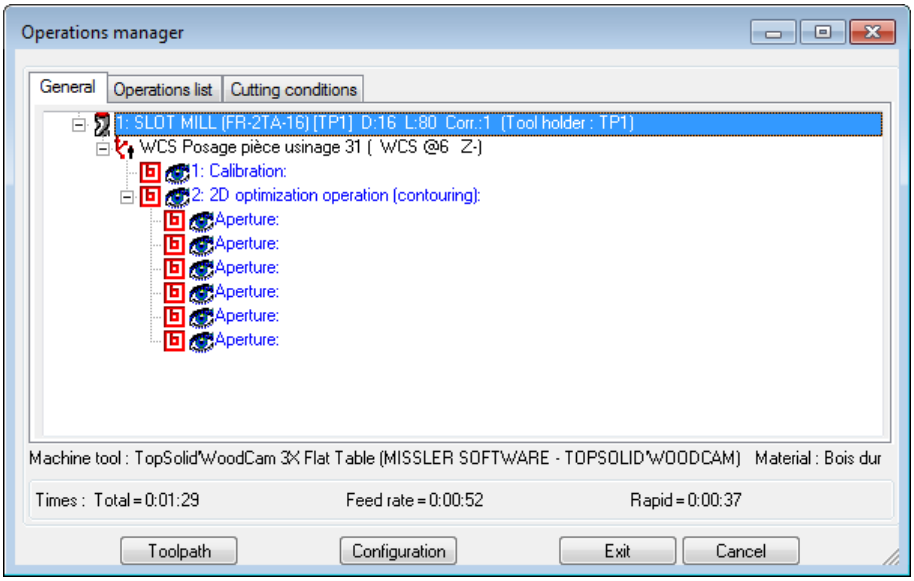


2D optimization of wood operations

2D optimization of wood operations is available on operations with closed path.
Operations with open path can be selected but optimization will only use the starting point of the operation.

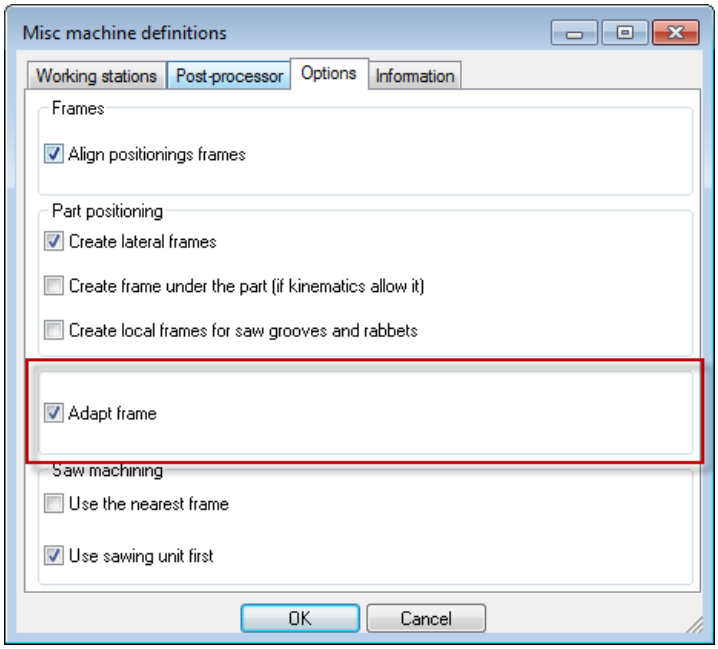






Adapt frame

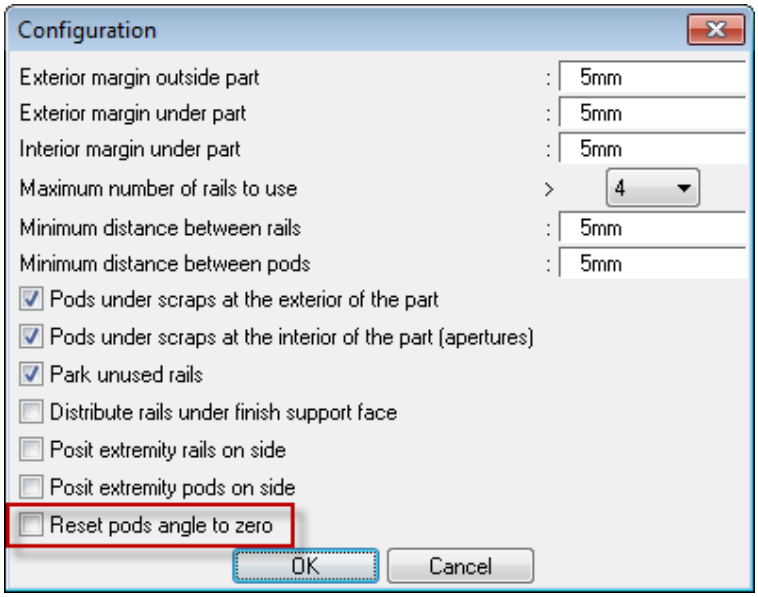
It is possible to run the **Work Coord. Syst. | Adapt frame** function after executing the operations performed with the **Wood machining | Operations analysis on a part** function.
This option is now the same as the one used for multi-machining.
You can manage this option in **Misc. | Machines | Misc. modifications**, and then in the **Options** tab by checking or not **Adapt frame**.



Rails and pods

Pod angle

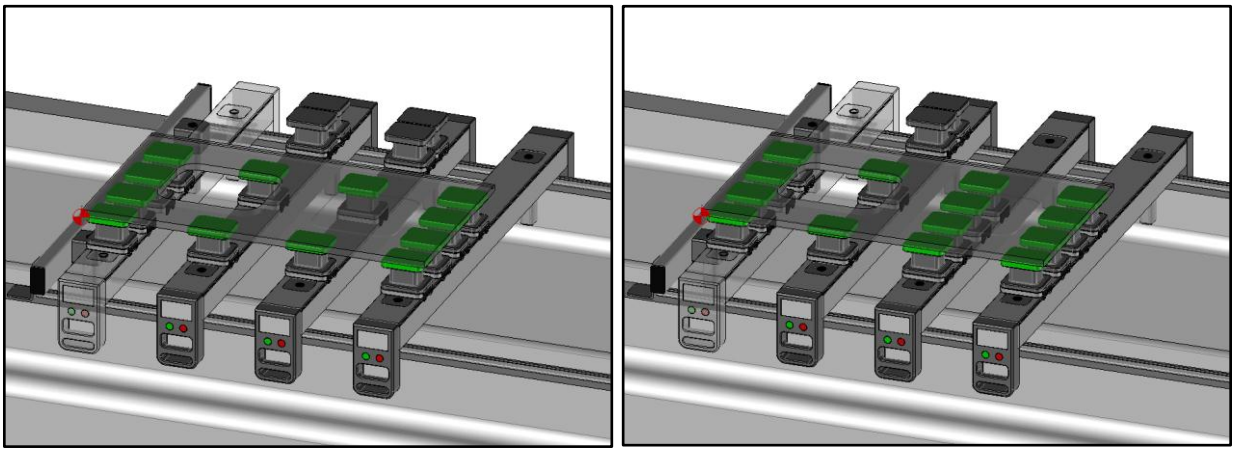
You can give an angle to a pod before performing automatic proposed positioning of rails and pods.
An option allows you reset all pods to 0° before performing the positioning.



Rail and pod positioning on first positioning face

Rail and pod positioning for the first machining position has been improved to optimize the pod positioning using only through operations.

Results in V6.13 and now in V6.14 with a pocket underneath:



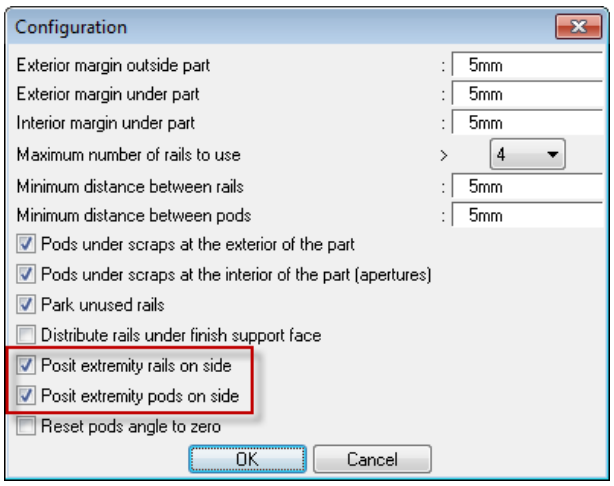
Rail and pod positioning on second positioning face

Rail and pod positioning for the second machining position has been modified using all operations and updated stock.

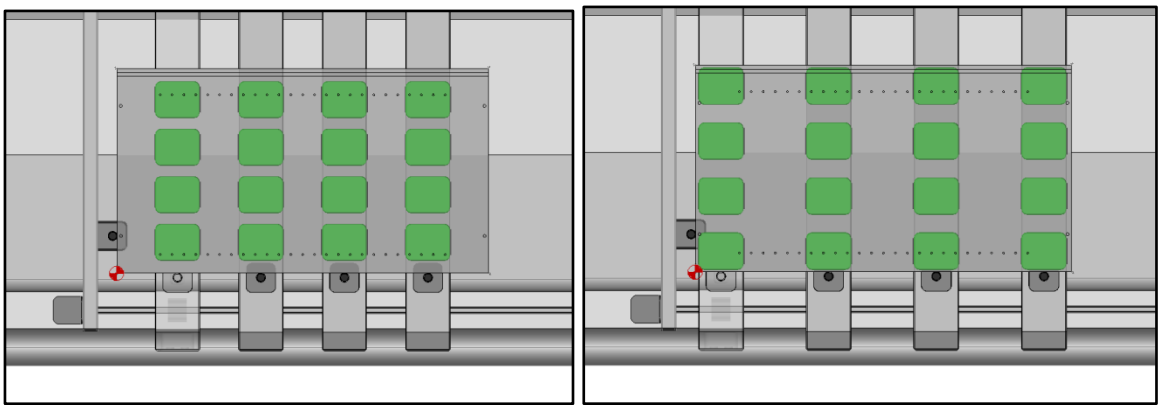
Rail and pod positioning on the periphery of the part

Two new options are available to optimize the rail and pod positioning on the periphery of the part.

These options are optimized for rectangular shapes.

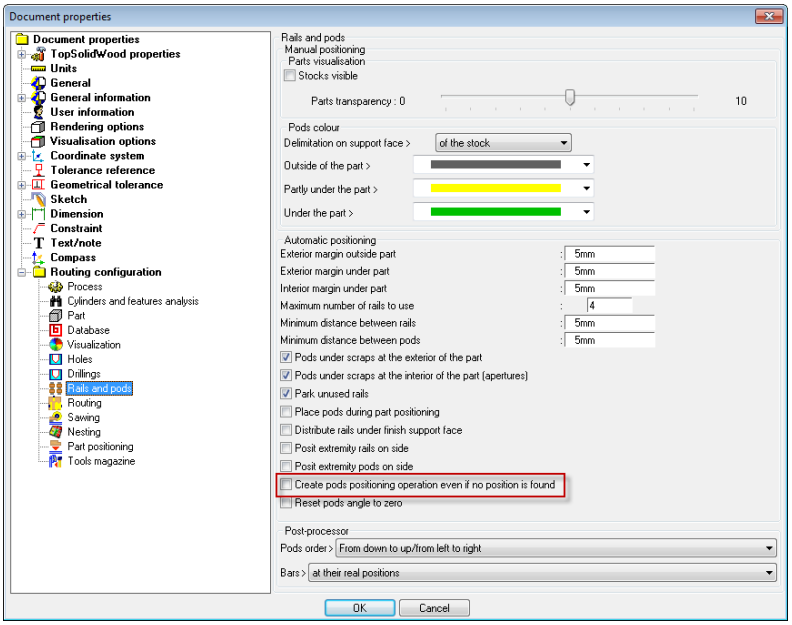


Results without and with these options:



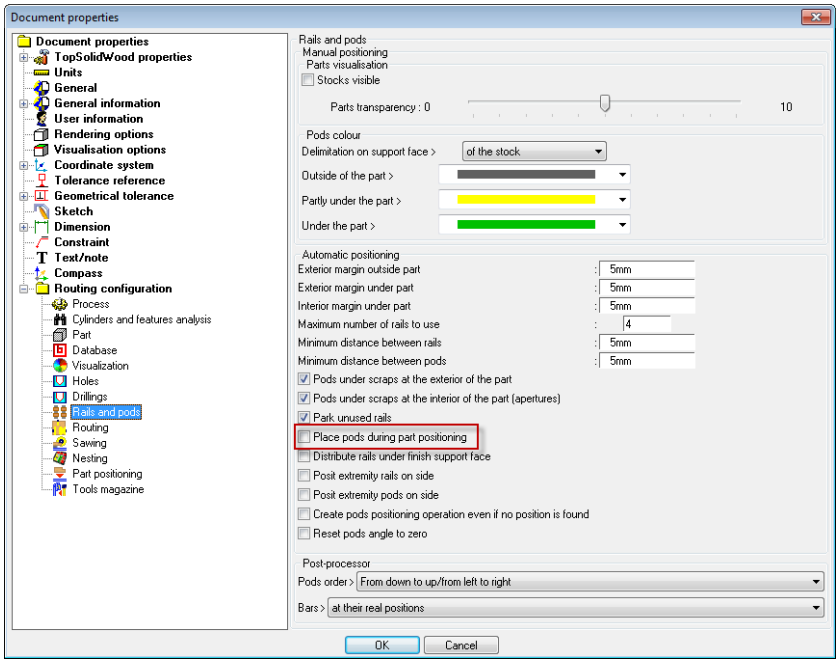
Creation of the rail and pod operation

An option allows you to create the rail and pod operation even if there is no solution under the part.



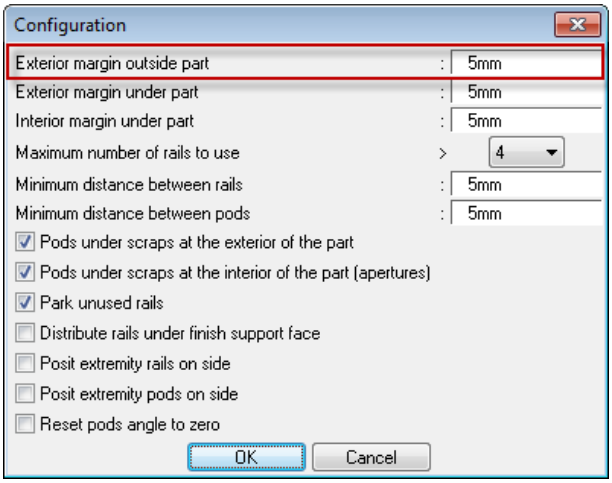
Rail and pod operation after turning over the part

A rail and pod operation is now automatically created after turning over part if the **Place pods during part positioning** option is checked.



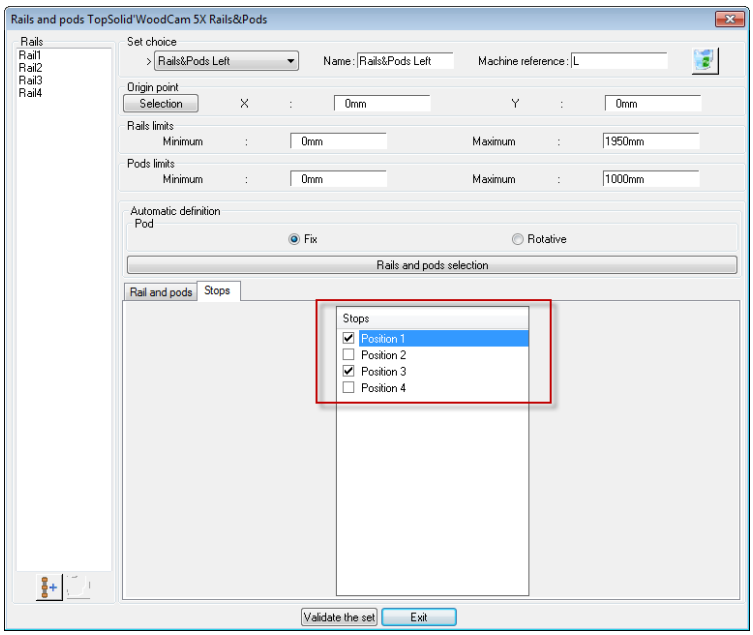
Outside margin for rail and pod positioning

An outside margin can be defined for pod positioning under the stock outside the finish part.



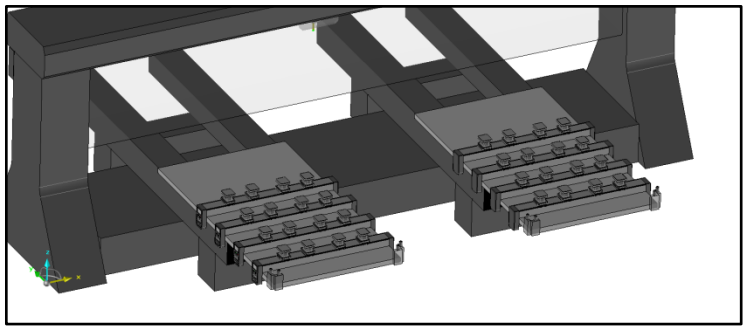
Choice of rail and pod set according to the used stop

For each rail and pod set, you can define the list of stops for which the sets will be used.
When automatically creating rail and pod operation, TopSolid'WoodCam will choose the good set of rails and pods depending on the stop used.



Rail and pod definition

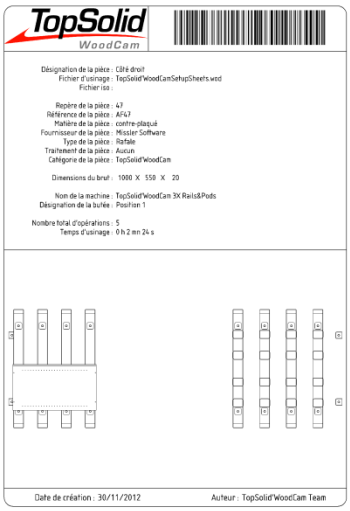
Rails along X axis can be defined using the rail and pod definition function.
No option is required; the orientation detection is automatic.

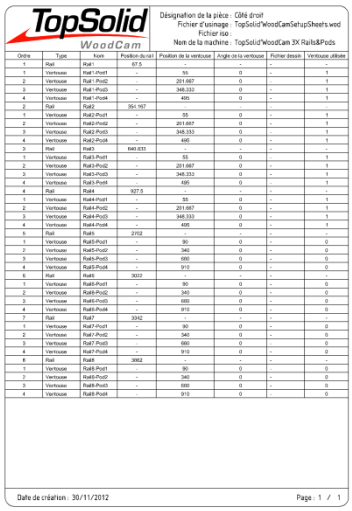


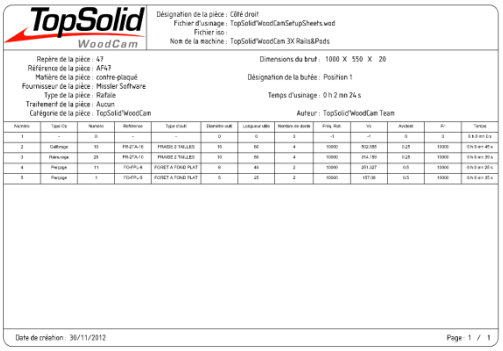
Setup sheets

New setup sheets

New TopSolid'WoodCam setup sheet documents have been added.

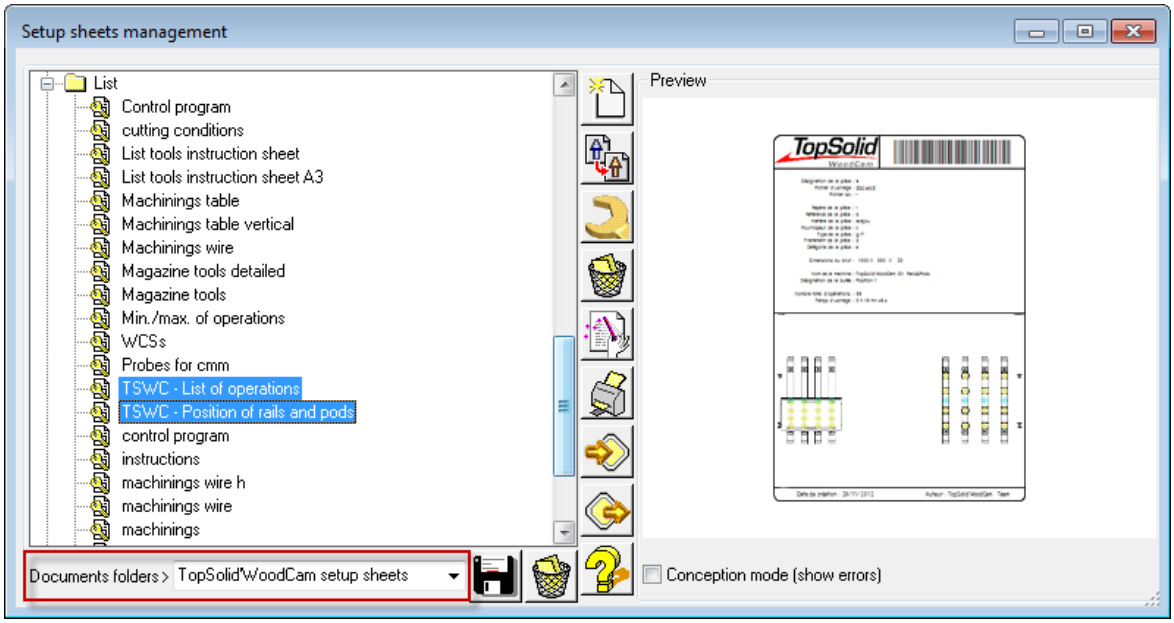


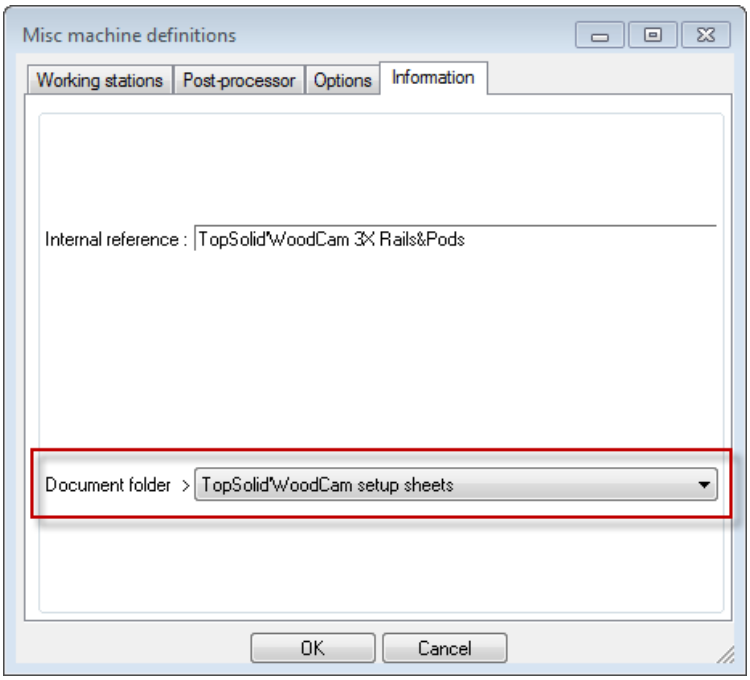




Setup sheets with multi-machining

Multi-machining allows you to create setup sheets for each part.
To do this, you have to create a setup sheet folder using the **Operations | Setup sheets** function and affect this folder to your machine template using the **Misc. | Machine | Misc. modifications** function, in the **Information** tab.



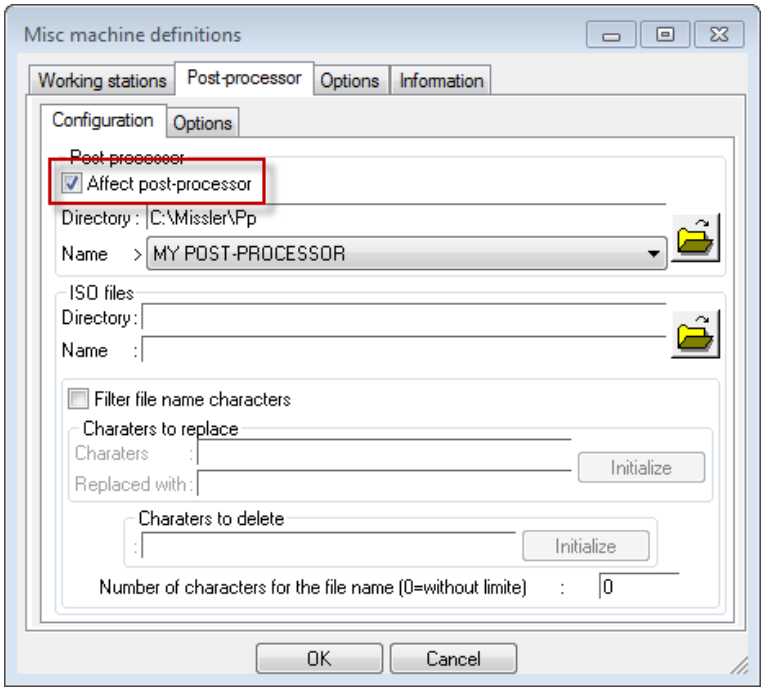


Post-processors

Post-processor assignment

A specific post-processor can be forced when manually generate the iso file as we already can do it for multi-machining. This option could be defined within **Misc. | Machine | Misc. modifications** in **Post-processor** tab.

This option is valid for both multi-machining and manual generation of the iso file.



New properties in the PDB

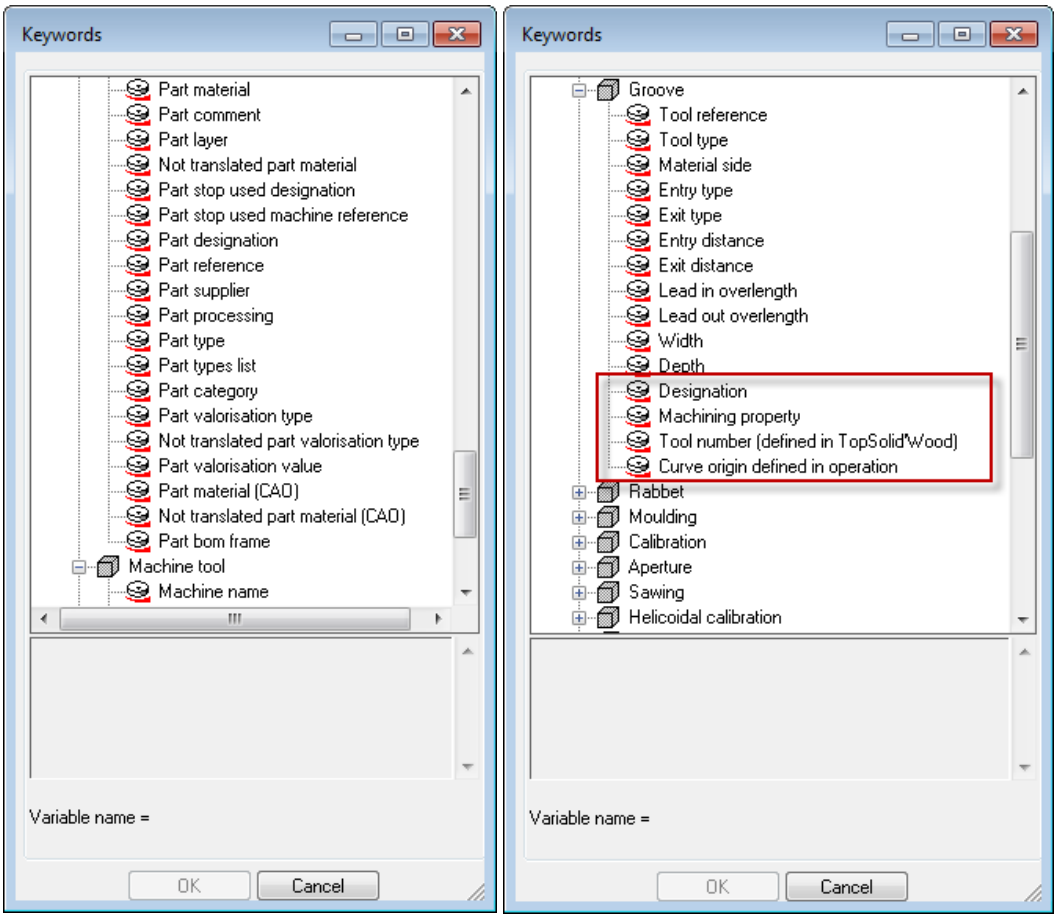
New variables are available within the PDB:

- Valorization value : **Part.valorisation_property_value**
- Total length of the tool with its attachment: **Tool.ZProg**

Processes

New properties for processes and setup sheets

New properties are available for processes and/or setup sheets to allow better and more advanced customization for machining.
These properties relate to wood operation properties, as well as part properties and machine properties.
A new type of draft view has been added in order to get TopSolid'WoodCam part view and a new table for rail and pod list is available as well.

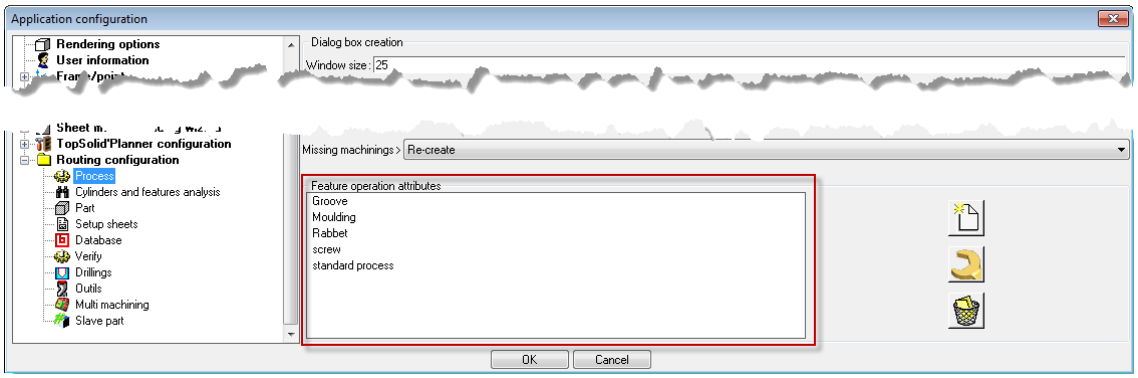


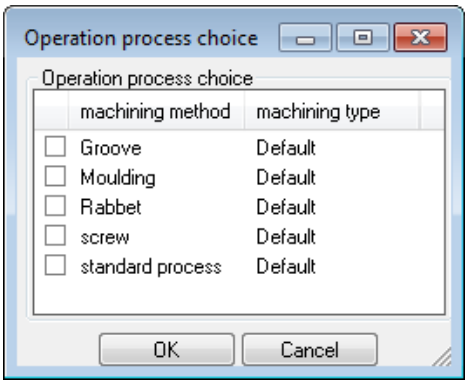
Comment and PP fields

The **Comment** and **PP** fields of the different operations are now associative with processes and will be re-executed as other fields already were.

Feature operation attributes

Lists are now sorted by alphabetic order in either options or process associations.



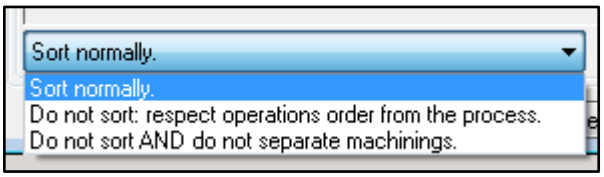


Sorting of process operation

An additional option is available to keep operations contained within a process as a single group.

These options are:

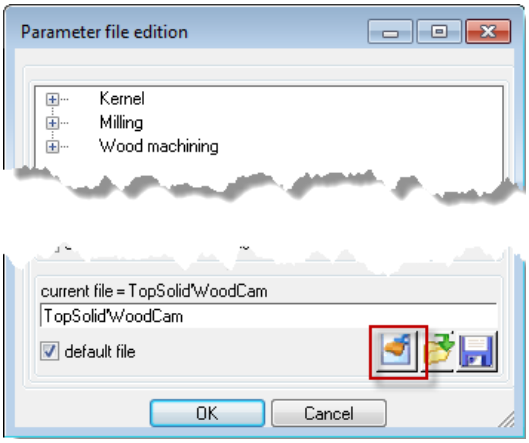
- **Sort normally**: The sort is done as if operations are independent of each other.
- **Do not sort: respect operations order from the process**: The sort will keep the operation order but operations will not necessarily be one after the other.
- **Do not sort AND do not separate machinings**: The sort will manage operations as a single operation so as to keep the order and execute operations one after the other.



Default values

Initialize default values

You can initialize default values to go back to the original configuration of TopSolid'WoodCam from the default value definition window.



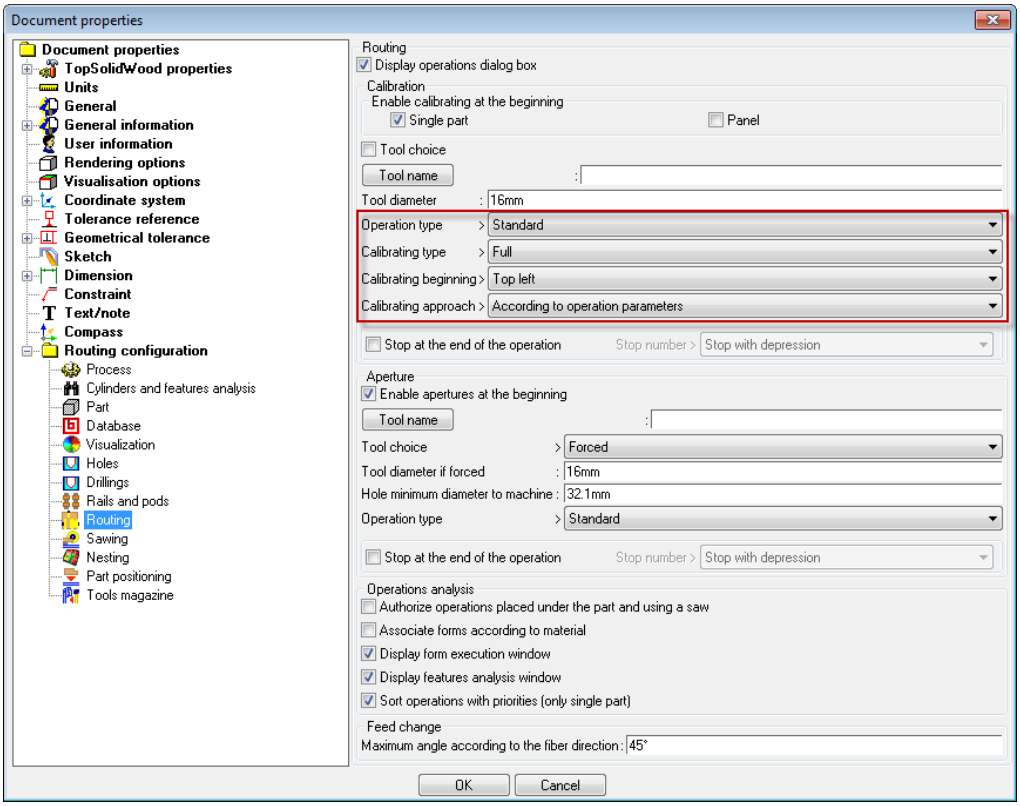
Options and properties of the machine template

Miscellaneous options of the machine

TopSolid'WoodCam dedicated options which were in the **Misc. | Machine | Modify locally** function have been grouped together in the **Misc. | Machine | Misc. modifications** function.

Calibrating options

Calibrating options defined in the template document properties have been reorganized for greater consistency between them.

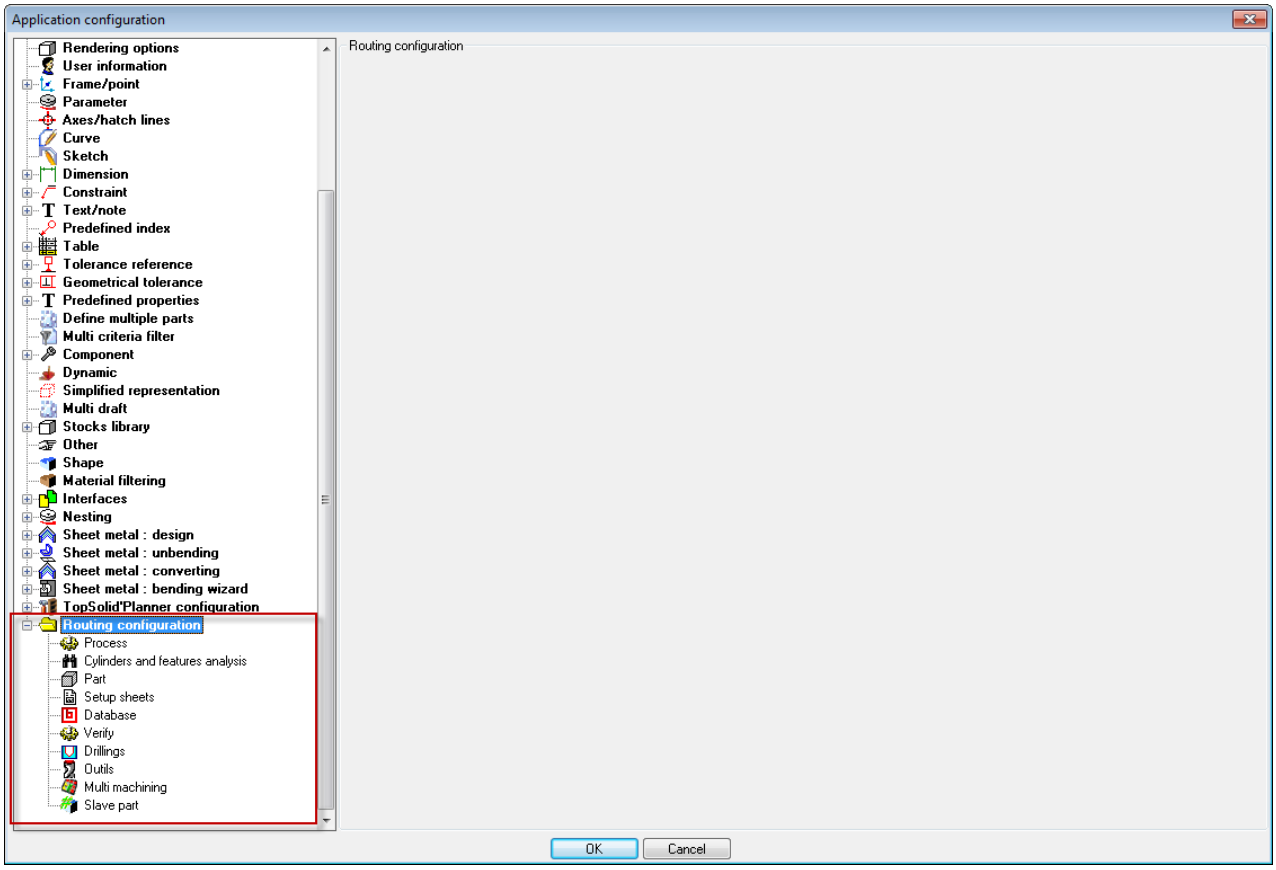


TopSolid'WoodCam options

All settings performed in the TopSolid'WoodCam options (**Tools | Options**) are now stored and read in the **topzwood.cfg** file only.

Simplification between main options and document properties

To avoid confusion between what is defined in the main options of TopSolid'WoodCam and what is defined in the properties of the document, the main options have been simplified.



TopSolid'Planner 2013: What's New

This document describes the improvements made to the **TopSolid'Planner** software: **2013** version.

Panel brush

The new panel brush allows you to change the panel type (one edge, four edges, with laminates…), change the support material, configure the edges (profile, matter, cuts and code) and configure laminates of a panel entity directly in TopSolid'Planner.

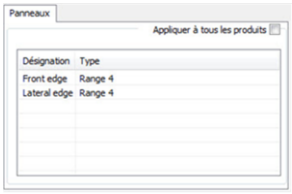
TopSolid'Planner project containing panel brushes.

Panel configuration by panel support and edges matter and coating modification.

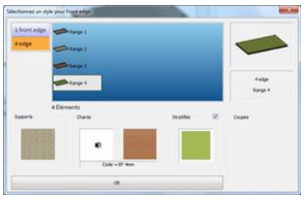
Panel brush overall structure

Planner component to configure

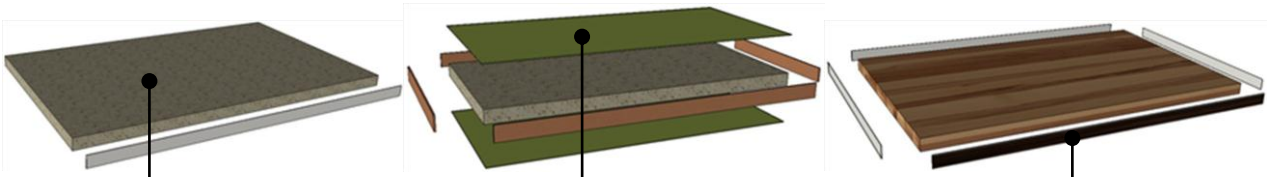
Brush choice to configure



Panel type choice and panel configuration

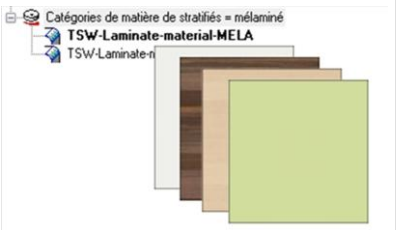


Panel categories containing different panel models

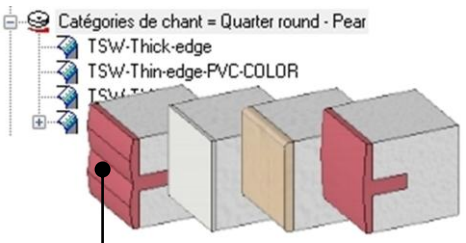


Support mater categories containing matters files

Laminate matter categories containing matters files

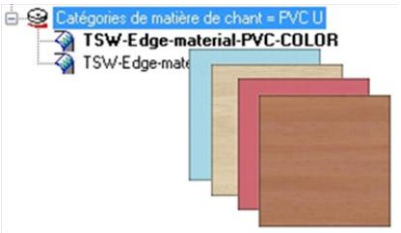


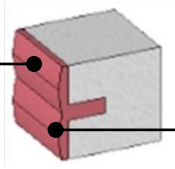
Edge categories containing different edge models



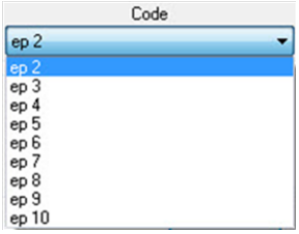
Edge model containing matters categories and a catalog

Edge matter categories which contain matters files





Catalog which contain different edge dimensions



Prerequisites prior to creating a panel brush

The panel brush, like the other brush types, requires the creation of panel categories described in the brush definition.

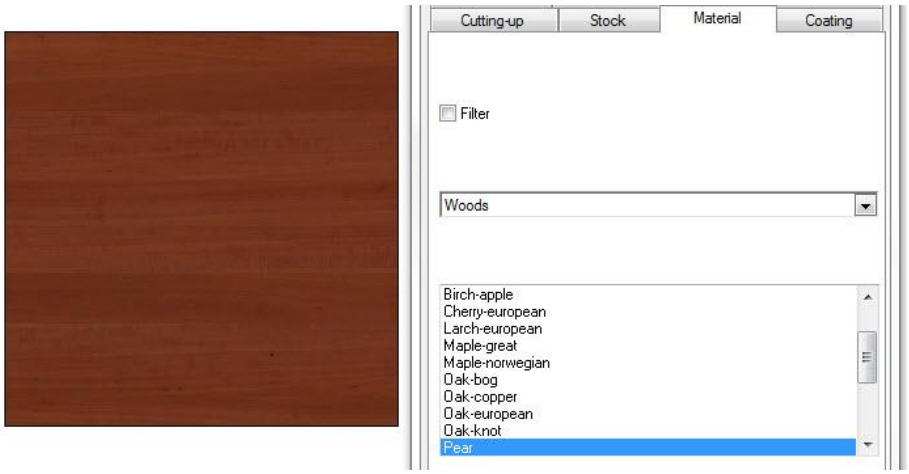
A panel is made up of a support, edges, laminates and options (cut type, laminates…). The support, the edge and the laminate should therefore have dedicated categories.

Creation of the matter and coating library

Creation of the matter and coating library for:

- The support;
- The edges;
- The laminates.

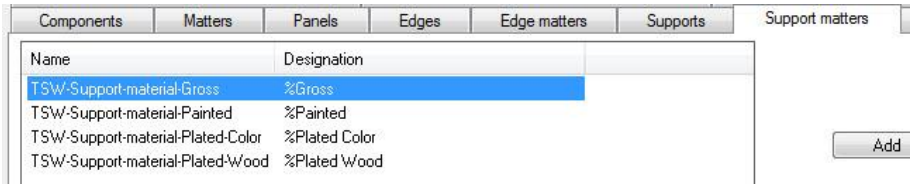
The process to create the edge, support and laminate's matter and coating categories is the same as the matter categories.



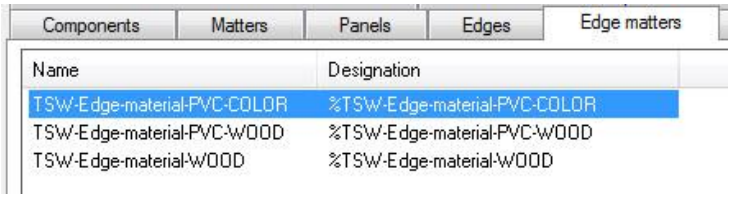
Planner's matter categories statement

Planner's matter categories statement in **Tools | Options | TopSolid'Planner configuration| Categories** for:

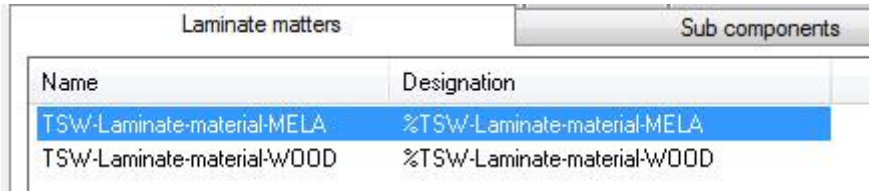
- Support's matter and coating



- Edge's matter and coating



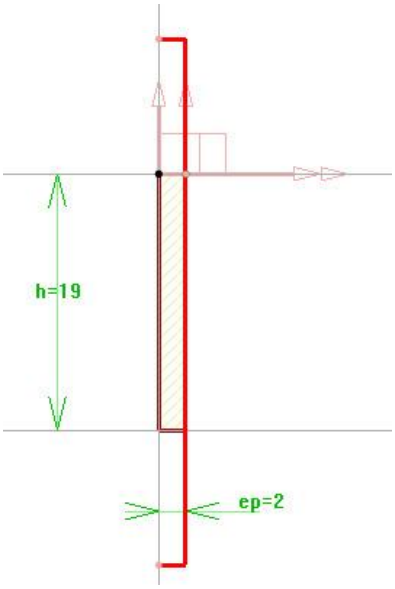
- Laminate's matter and coating



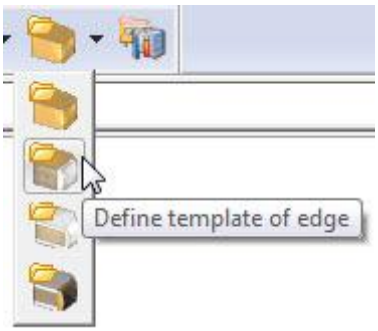
Edge's profile model creation

An edge model contains two data: the profile geometry (which can be managed by a catalog) and the matter's categories available for this profile.

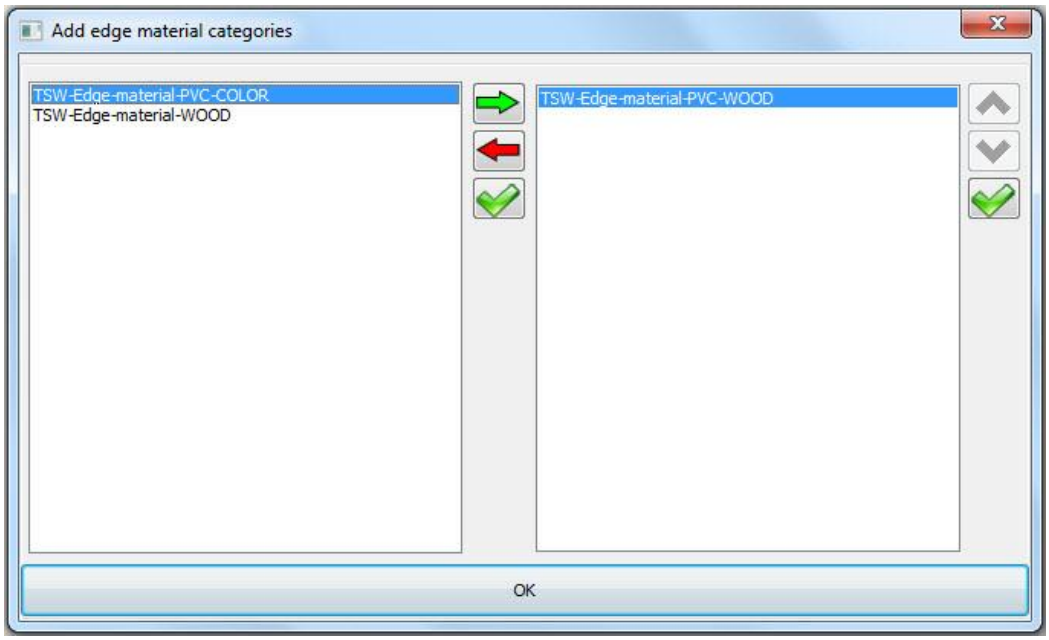
The edge's model creation starts by the standard TopSolid'Wood edge creation (see **TopSolid'Wood Help | Procedures | Creation of an edge model**). It is also possible to copy an existing edge model.



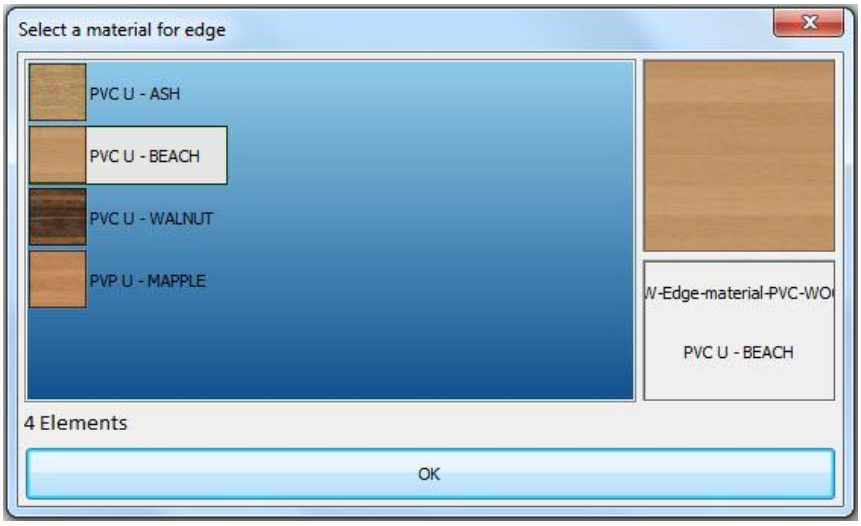
- Launch the edge's model creation function using the Planner menu, **Tools | Define model | Define edge model.**



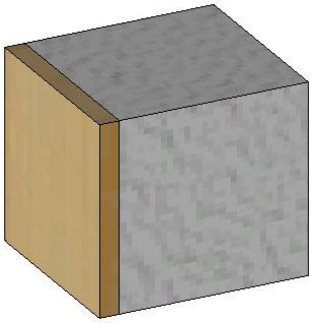
- Select the edge's matter categories that can be used by this edge profile.



- Then select the default matter to use in the chosen matter categories.

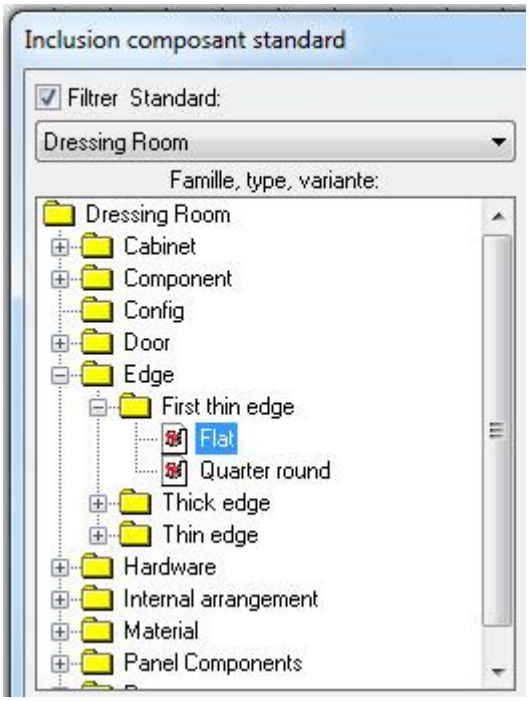


Then, the edge model is automatically generated. It remains only to save it in your component library.



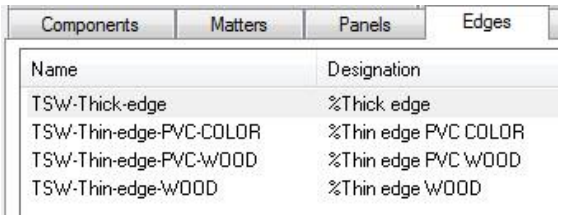
<u>Warning:</u> The edge model has to be saved in a library family named **edge** to be recognized during the codification generation.

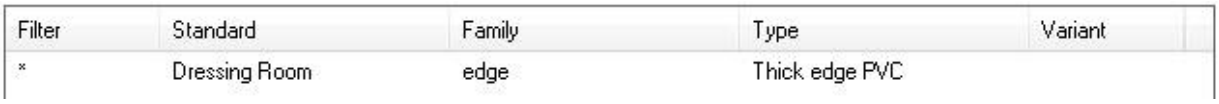
Moreover, the file **lib.cfg** located in the folder *Missler\V614\z\woo\lib\TOPWOOD* has to be copied in the library root folder.



Edge's categories statement

- Declare in **Tools | Options | TopSolid'Planner Configuration | Categories | Edge** the different categories previously created.

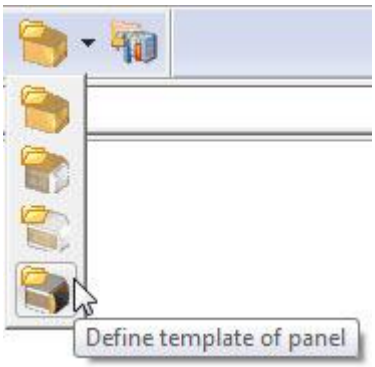




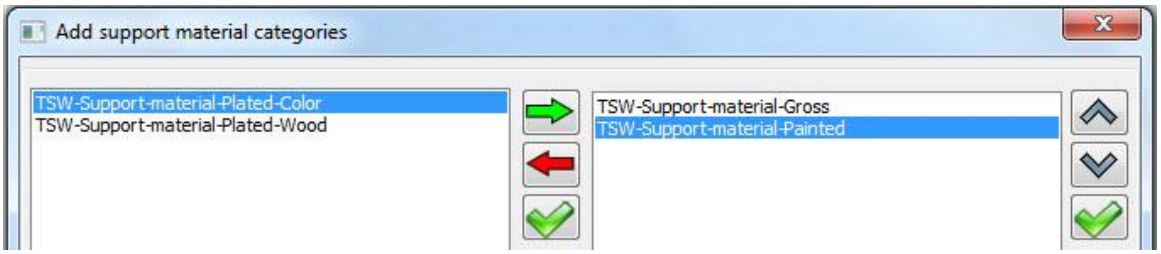
Panel model creation

Once the support's matter and the edge's model are created, it is possible to generate the panel model which will consolidate the different support's matter, edge and laminate.

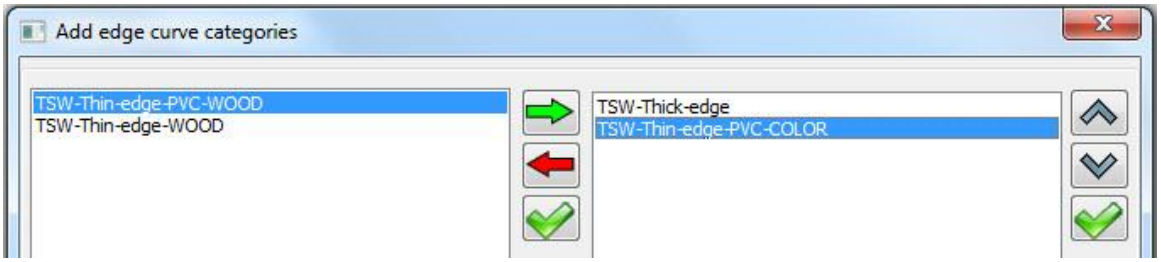
- Launch the panel model creation function using the Planner menu, **Tools | Define model | Define template of panel.**



- Choose the support's matter categories available for this panel and the default matter.

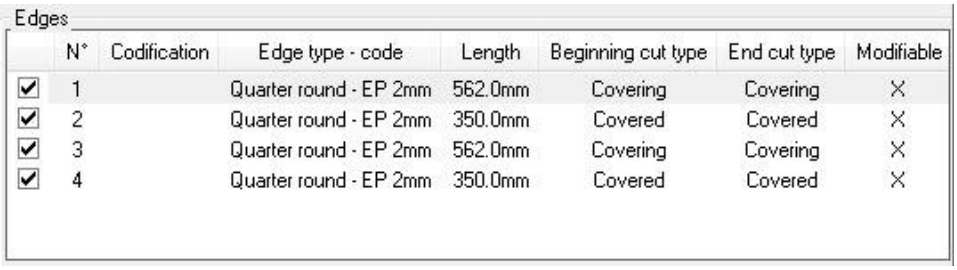


- Choose the edge profile's categories available for this panel and the profile, the code and the default matter.



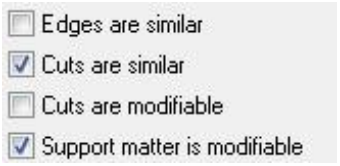
The panel model is automatically generated and the panel configuration window opens with different configurations areas.

Edge type



- **Checkbox**: Allows you to enable or disable the edge.
- **N°**: Edge number.
- **Codification:** Allows you to display the selected edge codification.
- **Edge type - code**: Allows you to display the selected edge type and code. By double-clicking on this case, it is possible to modify the edge.
- **Length**: Allows you to display the edge length of the model.
- **Beginning cut type/End cut type**: Allows you to set the cut type on the different edge.
- **Modifiable**: If the case is checked, it will be possible to configure this edge in TopSolid'Planner. All the checked edges will be identical after modification in TopSolid'Planner.

Edge configuration



- **Edges are similar**: Allows you to have four identical edges on the panel.
- **Cuts are similar**: Allows you to have the same cut type on the four edges.
- **Cuts are modifiable**: If this checkbox is checked, it will be possible to configure the edge's cuts type in TopSolid'Planner. The cuts type modified are the cuts of the edge checked **Modifiable.**
- **Support matter is modifiable**: If this checkbox is checked, the support's matter and coating can be modified in TopSolid'Planner.

Advanced options

As with the panel entity creation, the advanced options allow you to set the panel parameters.

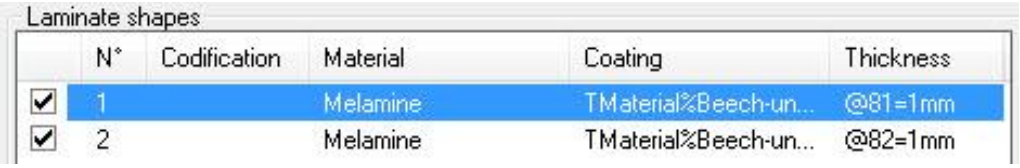
Here, it is strongly advised to use the panel **Simplified representation** to ensure the snap on panel's faces and their interchangeability.

Laminate types

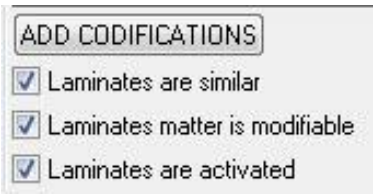
- To configure the laminate, first check **Make laminate** in the advanced options.

The part about the laminate is now available.

- Check a box to activate a laminate and choose the laminate's matter categories and the default matter.

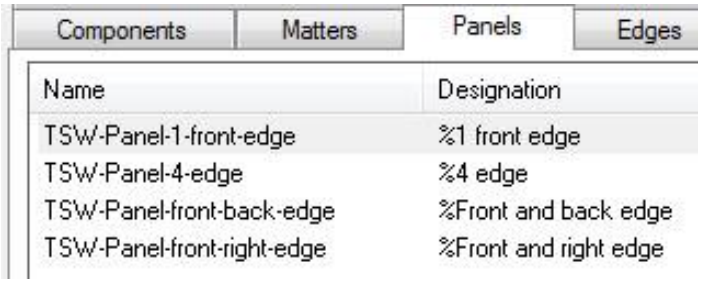


Laminate configuration



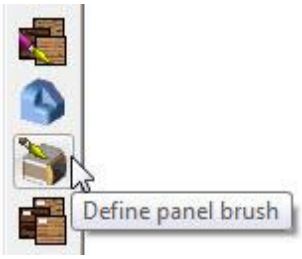
- **Laminates are similar:** Allows you to have two same laminates on the panel.
- **Laminates matter is modifiable:** If this option is checked, the laminate's matter and coating may be modified in TopSolid'Planner.
- **Laminates are activated:** This option allows you to disable the laminates during panel configuration in TopSolid'Planner.

- Save the panel model in your component library.
- Once the different panel models are created and saved in the library, declare the different panel's model categories in **Tools | Options | TopSolid'Planner Configuration | Categories | Panel.**

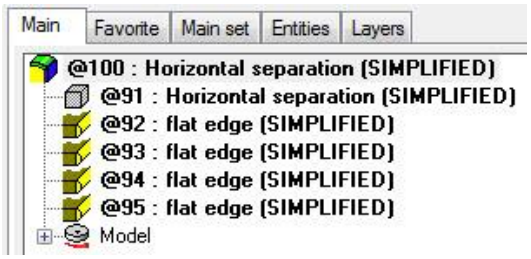


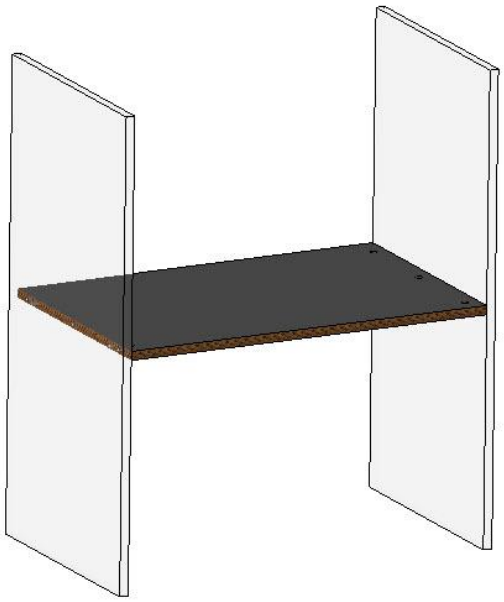
Panel brush creation

Because the panel brush only applies on panels, you must first create them. The options (support material, edges, laminates…) of the created panel are not essential because they will be then modified by the brush.

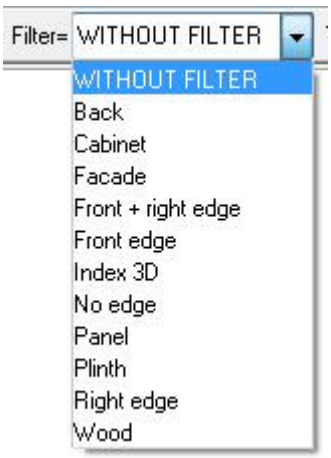


- Launch the brush panel function in the Planner menu, **Tools | Define panel brush**.





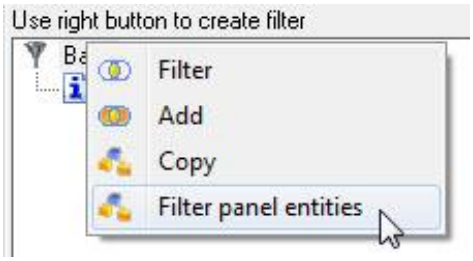
- **OK:** Allows you to validate the different panel brush parameters.
- **Designation:** Allows you to name the brush as it will appear in the TopSolid'Planner configuration. The different panel brushes are grouped by their designation and their categories.
- Panels selection : To select the different panels on which apply the panel brush, two ways are available :
    - Selection by **Filter**: Allows you to use a filter to select the panel.



In case this filter chooses the parts according to a property (type, designation…); it is important to apply this property to the **panel entity** and not to the **support**. To do this, launch the **Modify element** function on the panel entity, select **Panel Process,** and then **Modify Cutting-up.** Thereby, the property will be applied to the panel entity.

It is also possible to refine the filter by adding the panel entities:

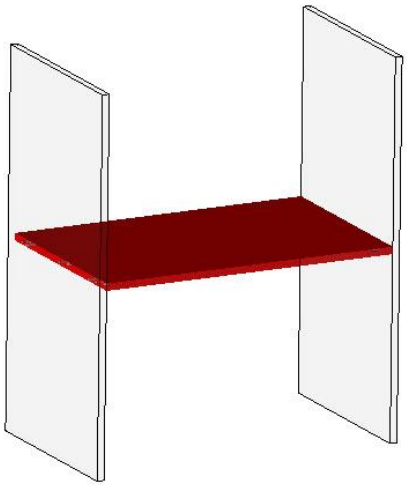
- In **Tools | Options | Multi-criteria filter**, right-click on the filter name and select **Filter the panel entities**.



  - Manual selection: if **No filter** is selected in the drop-down list, the **Panel(s)** option appears in the dialog bar.

It allows you to select manually several panels on which the panel brush will operate.

The different panels selected by the filter or manually are displayed in red.

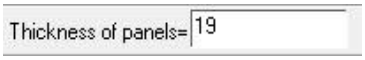


- **Template panel is modifiable = YES/NO**:
  - **YES**: It will be possible in TopSolid'Planner to change the panel model. It allows you for example to switch from a model with four edges to a model with one edge or with laminates…
  - **NO**: The panel model cannot be changed in TopSolid'Planner.

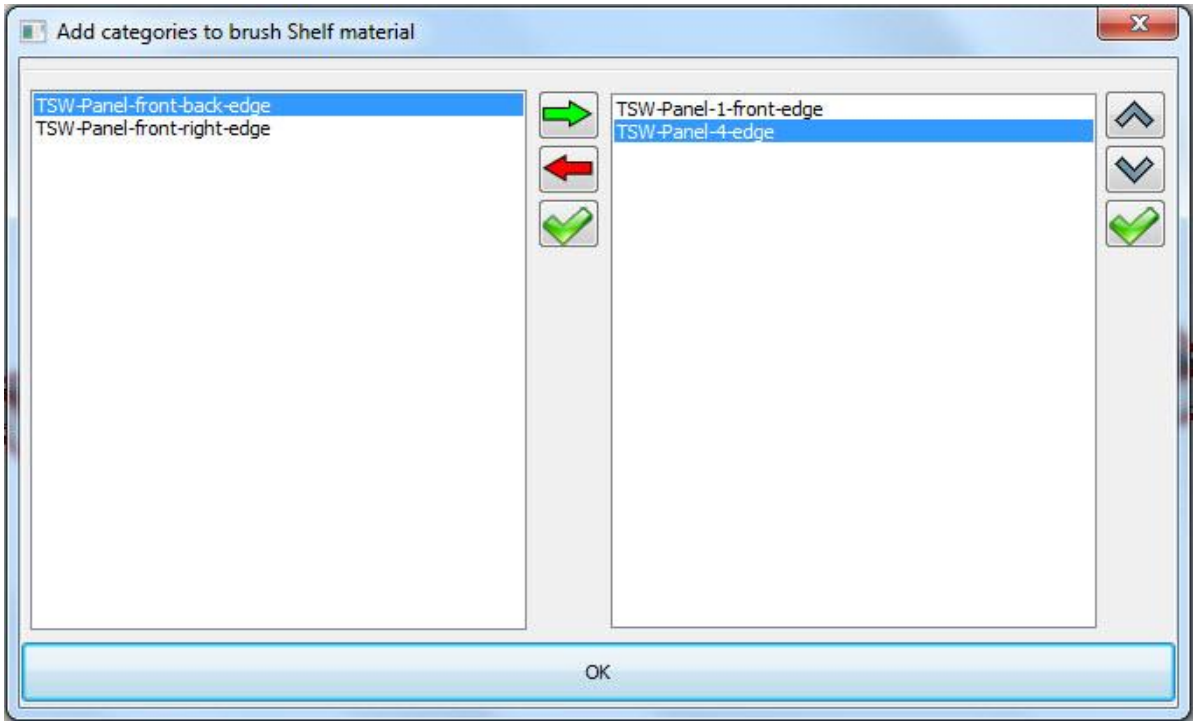
- **Multi-thickness = YES/NO**:
  - **YES**: The panel brush will modify all the selected panels, regardless of their thickness.

Using this option on **Yes** involves that all the panel categories using by this brush are available in all configurable thicknesses.

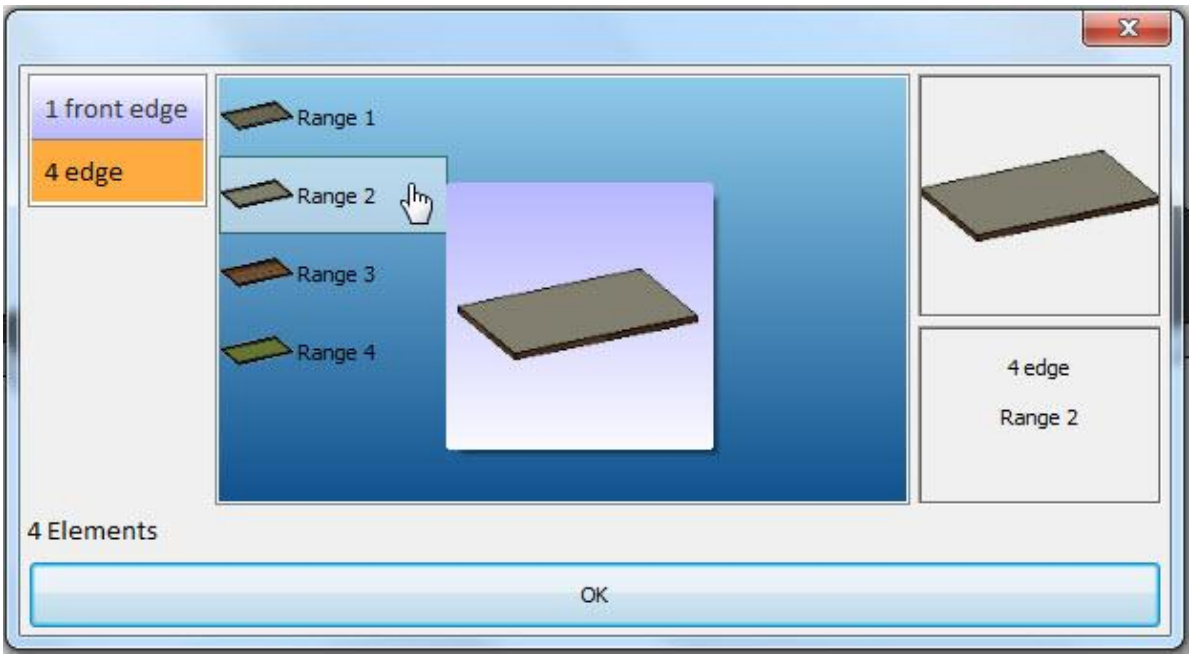
  - **NO**: Only the panels whose thickness is filled in the **Thickness of panels** field will be modified.



- Once all the information has been filled in, validate the brush with **OK.**
- Choose the panel categories which can be used by the brush.



- Then, choose the panel model used by default.

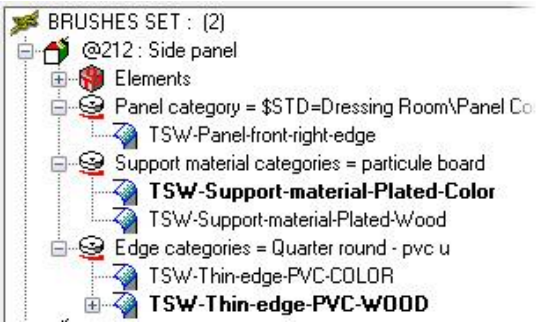


The brush is created and the different selected panels are configured as the chosen panel model.

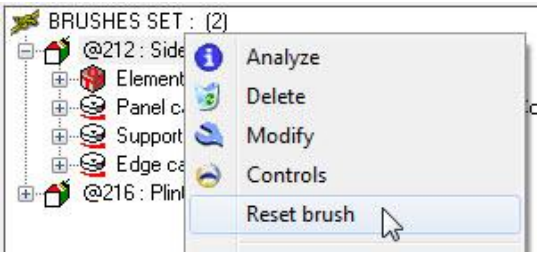
- Save the component.

Some data about the panel brush can be modified from time to time for this component from the tree.

- In the tree, **right-click | Edit set | Brush set**.
- Then on the category type to modify, **right-click | Modify.**



If the panel template used is modified after being used by the brush, it is possible to reset the brush with a **right-click** on the brush line **| Reset brush.**

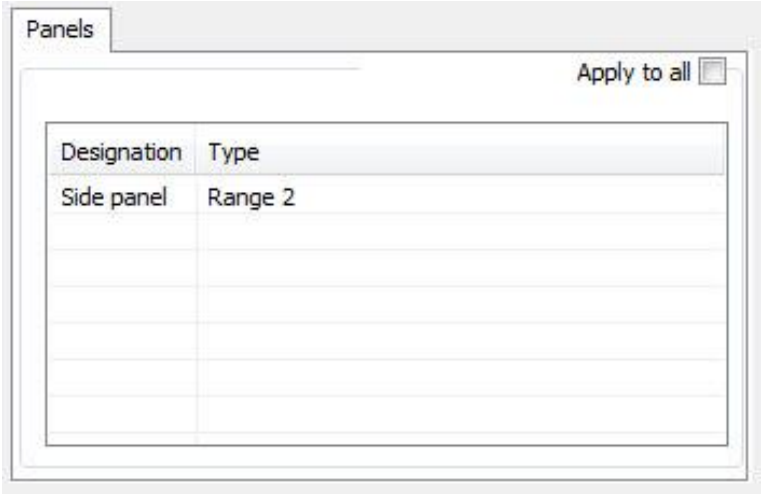


Using a panel brush in TopSolid'Planner

The panel brush, like the subcomponent brush and the material brush, is available when inserting and modifying a component.

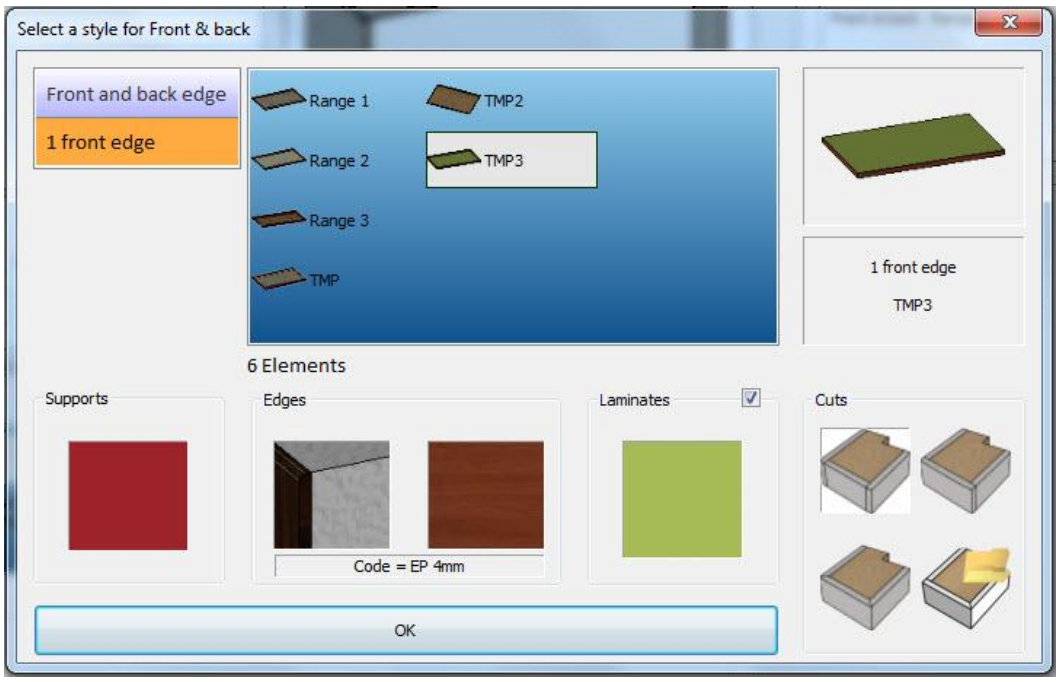
When configuring the component, the panel brushes are available in the **advanced mode.**

This tab available in the **advanced mode** allows you to configure the different component's panel brushes.



- **Apply to all**: Allows you to apply the chosen parameters to all document's equivalent brushes.
- **Designation**: Allows you to show the brush name.
- **Type**: Allows you to show the name of the panel template used.

By double-clicking on the brush line, the panel's configuration window opens.
It allows you to configure the panel depending on the chosen panel model and its parameters.



- **Panel model configuration:** Panel category choice and model selection. If during the panel brush creation it was informed that the panel model cannot be modified, this area is grayed-out.
- **Support:** Support's matter configuration. If in the panel model it was allowed to modify the support's matter, it is possible here to click on the preview to choose the support's matter in the different categories.
- **Edges:** Edges configuration. Allows you to simultaneously configure the different edges checked **Modifiable** in the panel model.
  If no edge has been checked **Modifiable**, these boxes are grayed-out.
- **Laminates:** Laminates configuration. Allows you to configure the laminates if the laminate modification was allowed. The checkbox allows you to activate or deactivate the laminates if the laminate deactivation was allowed in the panel model.
- **Cuts:** Cuts configuration. Allows you to set the edges' cuts types if the checkbox **Cuts are modifiable** was checked in the panel model.

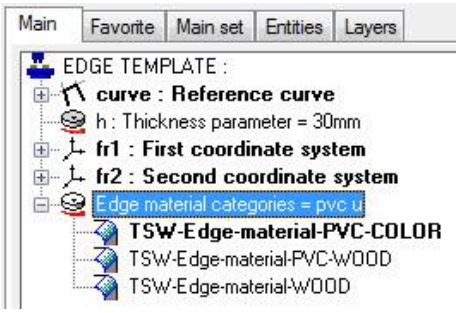
Advanced using

The advanced use of panel brushes allows you to drive:

- Support matter's categories in terms of their thickness;
- Edge matter's categories in terms of their dimensions (height and thickness),
- Laminate matter's categories in terms of their thickness.

Edges matter's categories driving

- In the edge's model file, edit the edge's model in the tree by launching the **Define model | Define edge model** command.



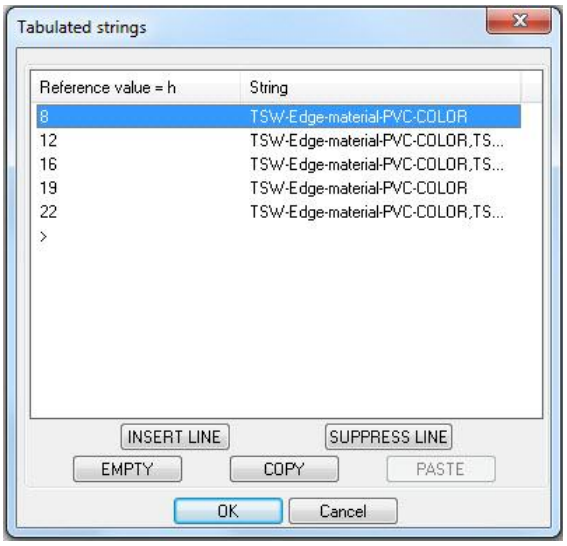
- On the **Edge's matter category** line, **right-click | Modify**.

Four options are then available:

  - **NO TEXT**: This option corresponds to the simple mode; the matter categories do not change depending on the dimensions. After selecting this option, choose the matter categories and the default matter.

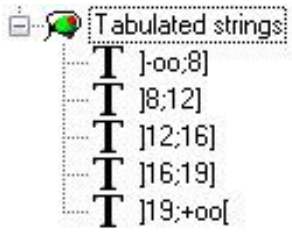
  - **TABULATED TEXT**: This option allows you to drive the edge matter categories according to its height (parameter h in the edge which corresponds in the assembly to the panel thickness).

The **Tabulated strings** window opens.



- In the first **Reference value = h** column, fill in the different values which will establish the intervals.

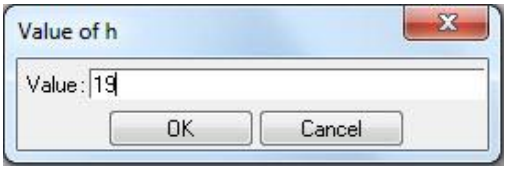
In the example shown here, the established intervals are:



- In the second **String** column, double-click on the line to select one or several categories to use for this interval.

  - **TABLE TEXT:** This option allows you to drive the edge matter categories according to its dimensions (parameters h and t in the edge).

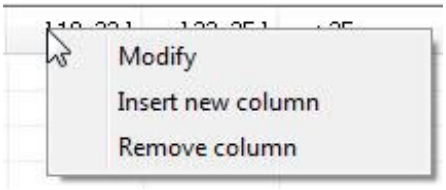
After selecting this option, a window opens with a first **Value of h** field; it corresponds to the first **h** value for the first height interval.



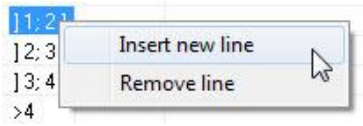
In the displayed table:

- The lines correspond to the different edge's thickness intervals;
- The columns correspond to the different edge's height intervals

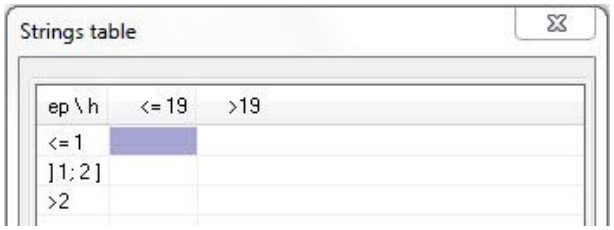
- To modify, add or delete an interval, right-click on the column head.



- To insert or to delete a line, right-click on the line head.



- To add edge's matter categories which correspond to a thickness and a height interval, double-click on the corresponding case.



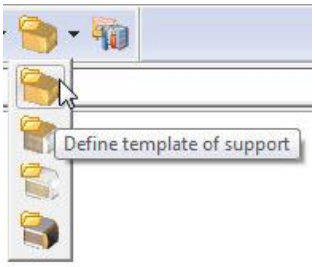
In the example above, the categories loaded in the selected case will be available when the edge thickness will be less or equal to 1mm and when its height will be less or equal to 19mm.

  - **Text**: For advanced users, it is possible here to select a text which contains an expression to drive the categories with more parameters and options.

- Once the edge matter categories are driven, save the edge template in the library.

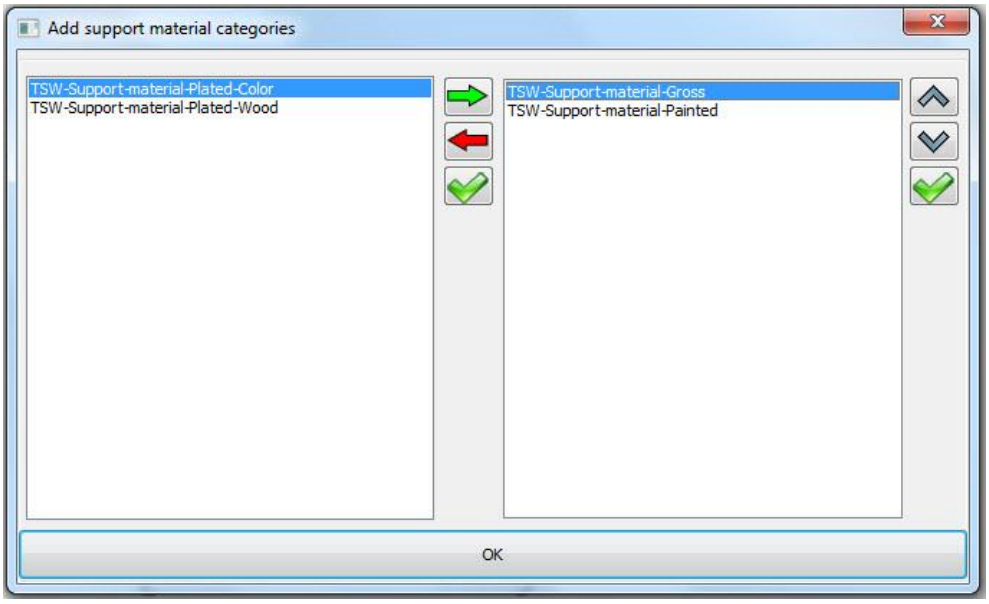
Panel support matter categories driving

The panel support matter categories driving starts by the support template creation with the Planner menu, **Tools | Define template | Define support template.**



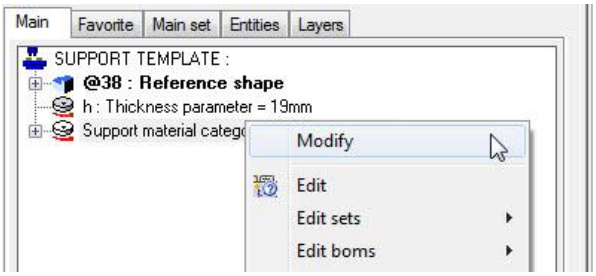
- Choose the support matter categories by default and the default matter.

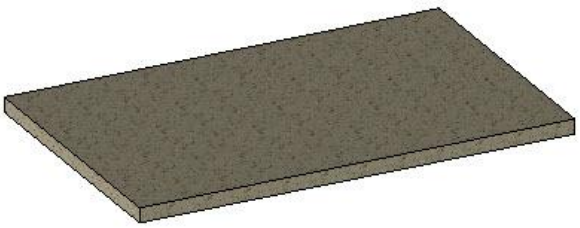
The chosen categories will be then modified by the driving.



The support template is then automatically generated and the tree opens on its parameters.

- On the **Support material categories** line, **right-click | Modify**.





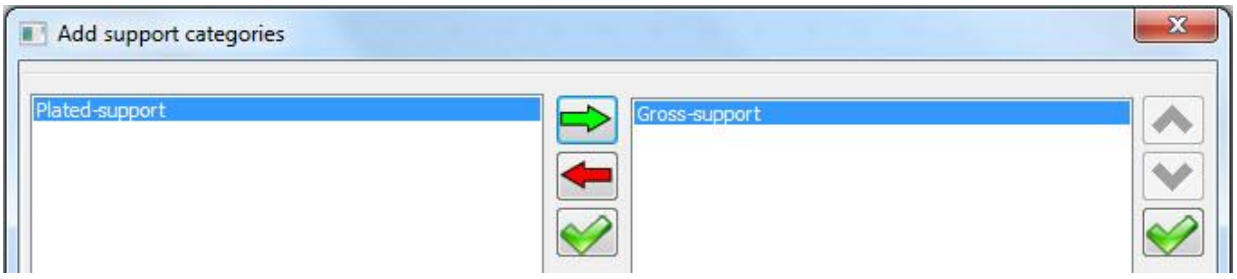
Three options are available in the dialog bar:

  - **NO TEXT:** Allows you to drive the matter categories easily, without thickness condition. In theory, if the support matter categories are not driven with the **Tabulated text** function, the creation of a support template is not useful.
  - **TABULATED TEXT:** This option allows you to drive panel support matter categories according to its thickness (parameter h in the support template). The settings work in the same way as the **Tabulated text** in the edge template.
  - **Text:** For advanced users, it is possible here to select a text which contains an expression to drive the categories with more parameters and options.

- Save the support template in the standard library.

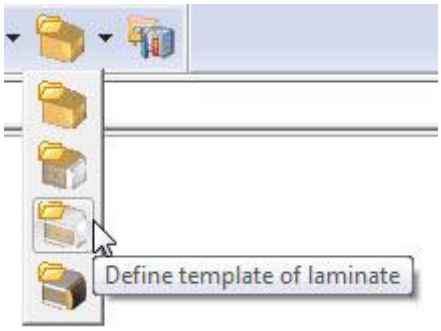
The different panel support categories created have to be declared in **Tools | Options | TopSolid'Planner configuration | Categories.**

With these settings, when a panel template is created, the support template categories will be required, as well as the default matter.



Laminate template creation

Creating a laminate template allows you to drive the laminate's matter categories according to its thickness.



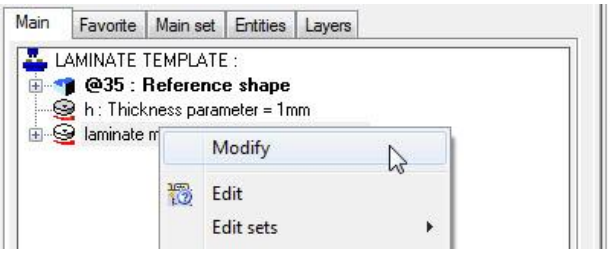
- Start by creating a laminate template with the Planner menu, **Tools | Define template | Define template of laminate.**
- Choose the matter categories and the default matter.

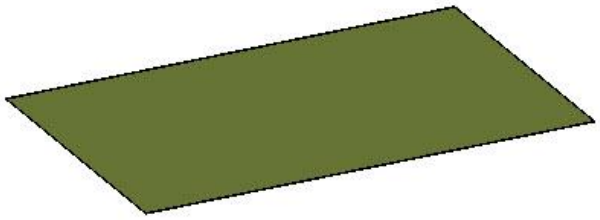
The chosen categories will be then modified by the driving.

The laminate template is automatically generated and the tree opens on its parameters.

- Save this new laminate template in the standard library.

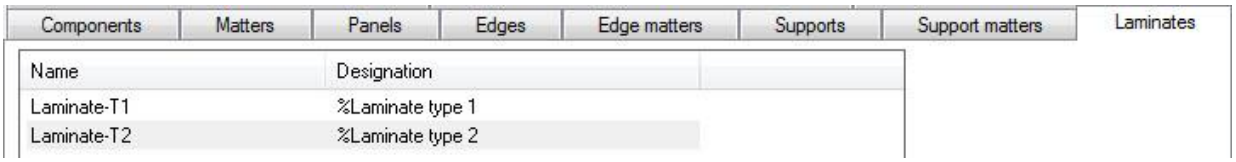
- In the tree, on the matter categories line, **right-click | Modify.**





The laminate matter categories driving works in the same way as the support matter categories driving.

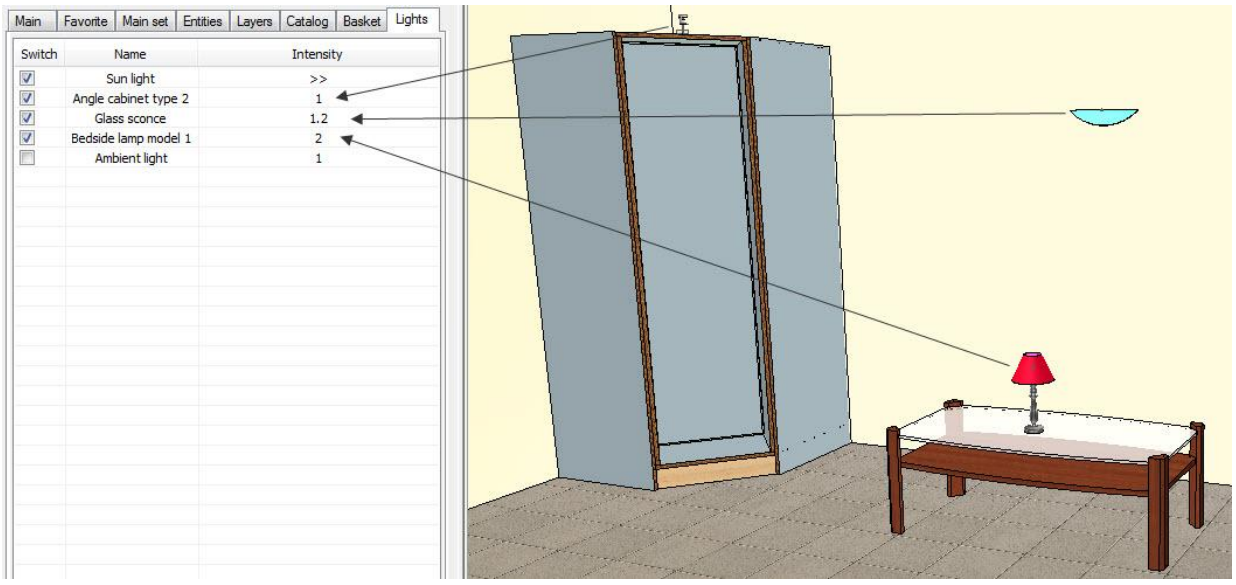
The different created laminate categories have to be declared in **Tools | Options | TopSolid'Planner Configuration | Categories.**

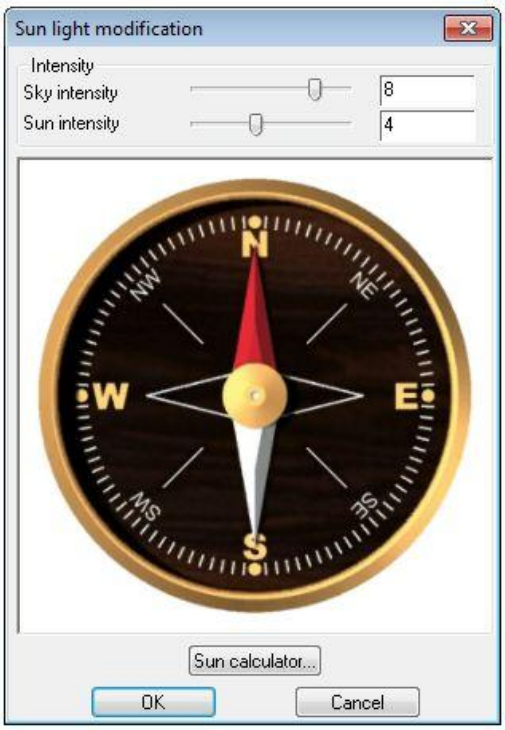


With these settings, when a panel template is created, the laminate template categories will be required, as well as the default matter.

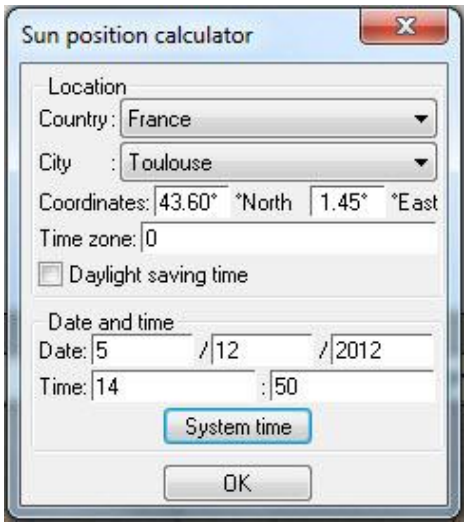
Light management in TopSolid'Planner end user

In TopSolid'Planner, a new interface allows you to manage very easily the Planner document's lights.
The **Lights** index gives a direct access to light activation/deactivation and intensity settings. The light list is updating when some components with light are included or deleted.
Each line corresponds to a component included in the Planner which contains one or several lights. The line name corresponds to the name of the included assembly which imports lights.
Switch on or switch off the line or switch on or switch off all the lights contained in the assembly.
It is also possible to vary the light intensity.





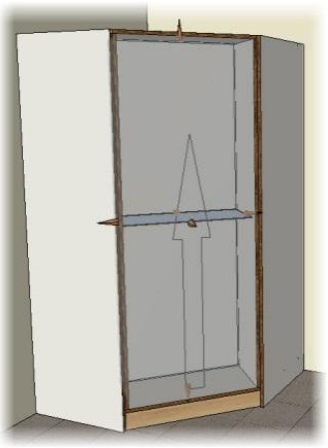
A dynamic modification of the sky and the sun is now available by clicking the **Sun light** line. It allows you to set the global lighting of the document and calculate the sun position using the **Sun calculator** function.



Elements visualization

The new function **Elements visualization** allows you to hide some elements included in a TopSolid'Planner document.

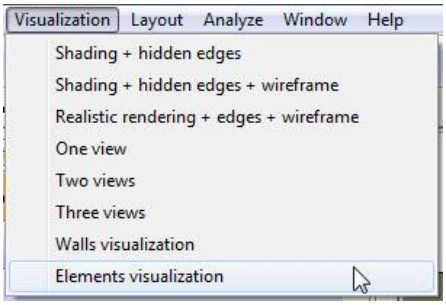
It allows you for example to momentarily hide a door to place a shelf behind.



This function is available in the **Visualization| Elements visualization** menu.

- **Mode = HIDE**

The **Hide** mode allows you to select shown elements to hide them.

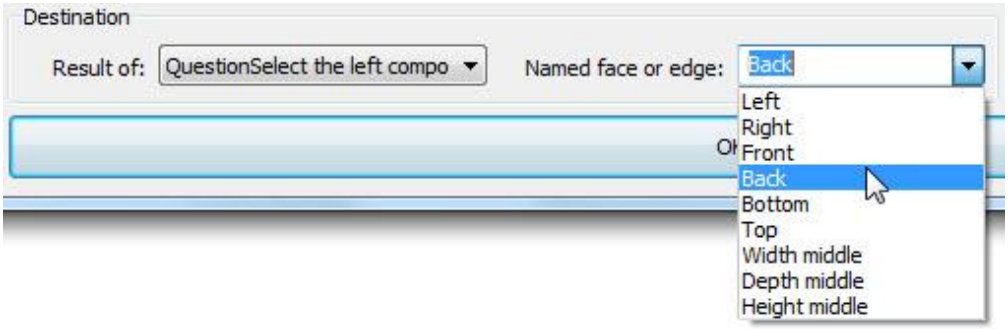


- **Mode = SHOW**

The **Show** mode allows you to select hidden elements to show them.

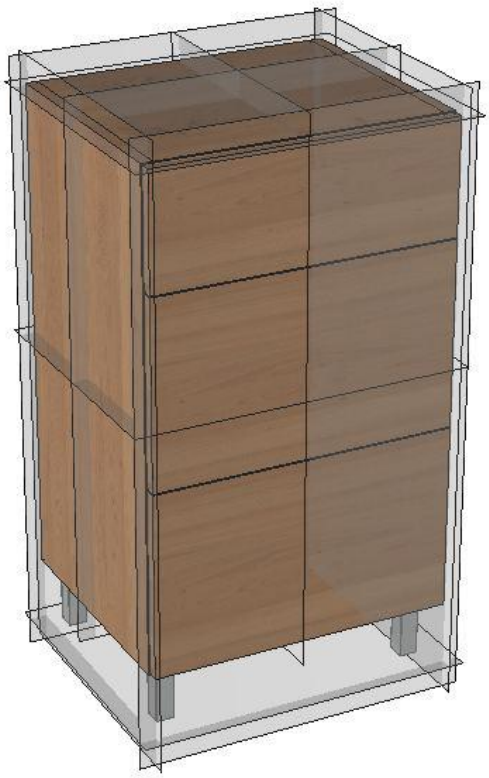
Improving face's name entering

In an element's positioning sequence creation, during the face's name entering, a drop-down list is now available. It contains the names of the publishings defined in **Tools | Options | Components | Components management.** This makes it easier to enter the name of the face and reduces the positioning errors due to a false publishing's name.



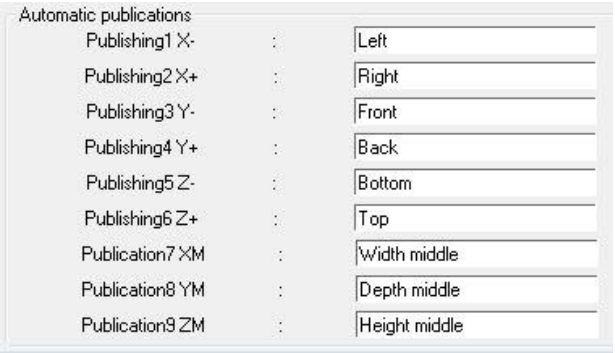
Centering constraint in positioning sequence

Now it is possible to create a centering constraint in a positioning sequence.
The **Automatically publication creation** function has been enhanced with the **Median publications** option. The names of these new publications can be adjusted with the other publication's name in **Tools | Options | Components |Components management.** These planes are placed in the middle of the two other publications of the direction.



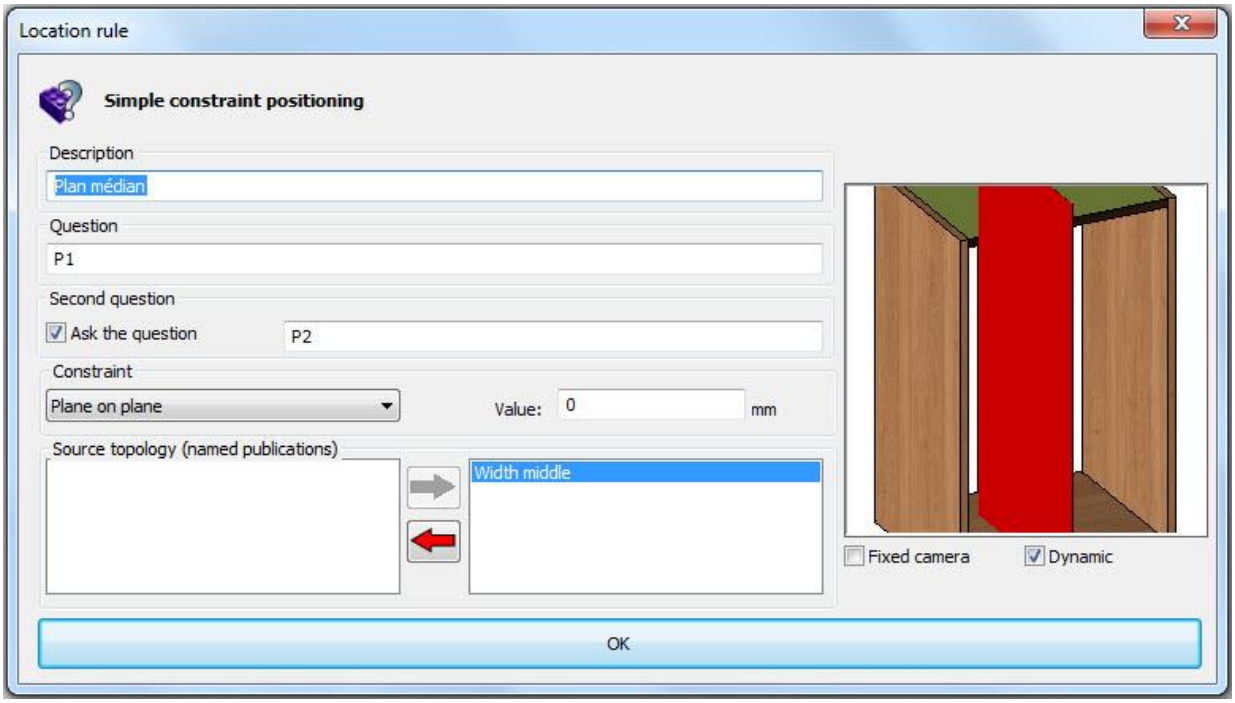
Of course, it is possible to create the publishing on an alternative set and on the current coordinate system.

In this example, the normal publishing is placed on the main assembly (all the elements), but the median publishing is placed only on the cabinet (without foots and the facades).



When adding a rule in a positioning sequence:

- In an implicit constraint: to place a median publication on a selected component's publication, it is possible to use the **Mate or alignment** constraint. This operation is similar to the positioning in older versions with the new possibility to use the median publishing.
- A new positioning constraint had been created in the simple constraint positioning: **Plane on plane** constraint with a **Second question** checkbox which gives the possibility to ask a second question in the same positioning constraint.

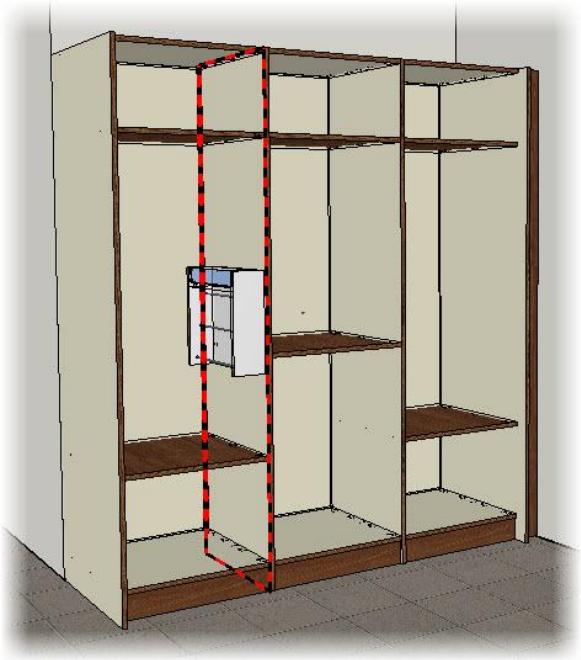


In this example, the **Width middle** publishing will be placed on the plane located between the two faces selected in the questions **P1** and **P2**.

Improving positioning filter

Some improvements have been made to the Planner positioning filter.

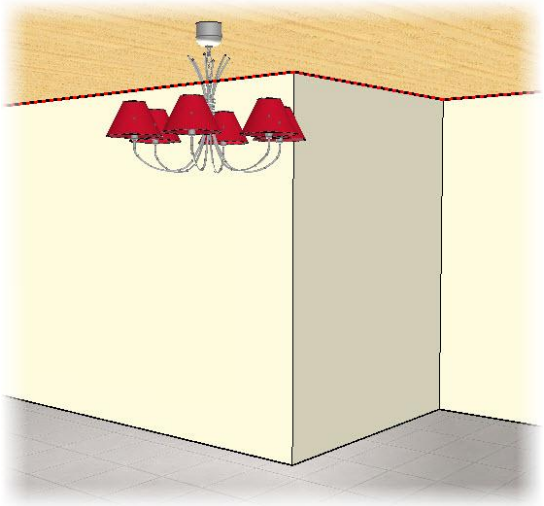
- Possibility to filter the **Vertical** and **Horizontal** component's faces in a positioning.
  For the **Horizontal** faces, it is possible to refine the filter by filtering the **Bottom** and the **Top** face.



*Positioning example with **Vertical** faces filter.*

*Positioning example with **Horizontal Top** faces filter.*

- A **Ceiling** filter has been added. It works the same way as the **Ground** and the **Wall,** but it allows you to select the ceiling.

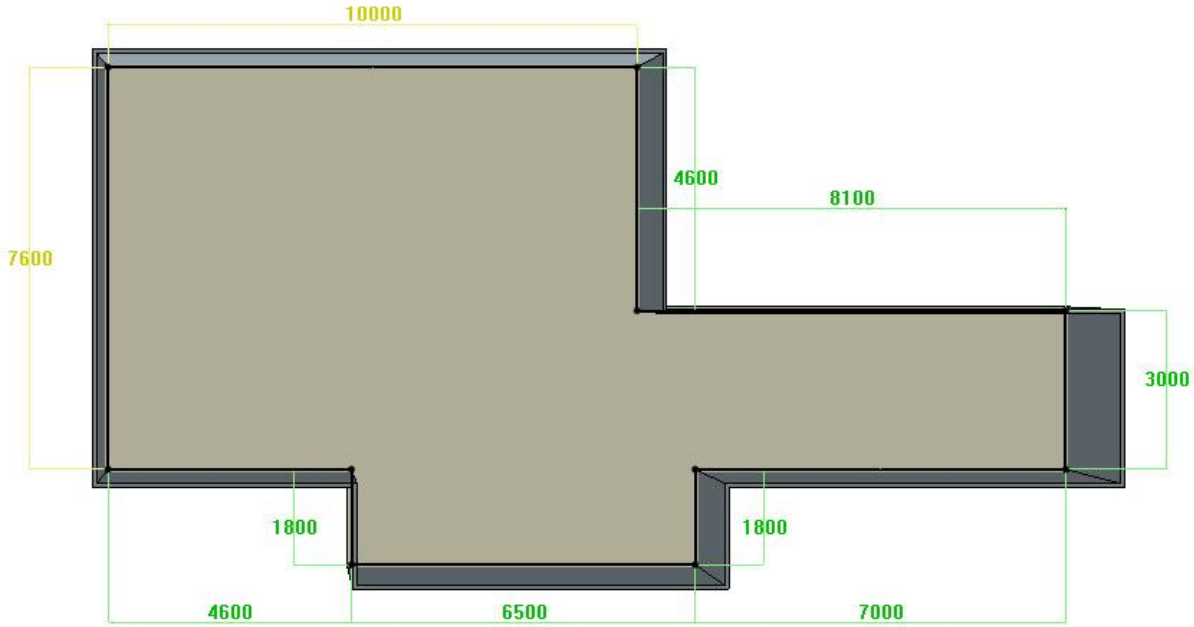


Version number in TopSolid'Planner Client

It is now possible in TopSolid'Planner Client to know the TopSolid'Planner version number (created by the administrator during generation) and the TopSolid core version.

Wall dimensioning improving

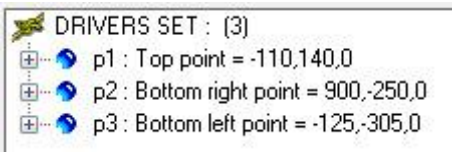
The wall dimensioning has been improved to makes it easier to modify. Moreover, the over dimensioning is now better managed.



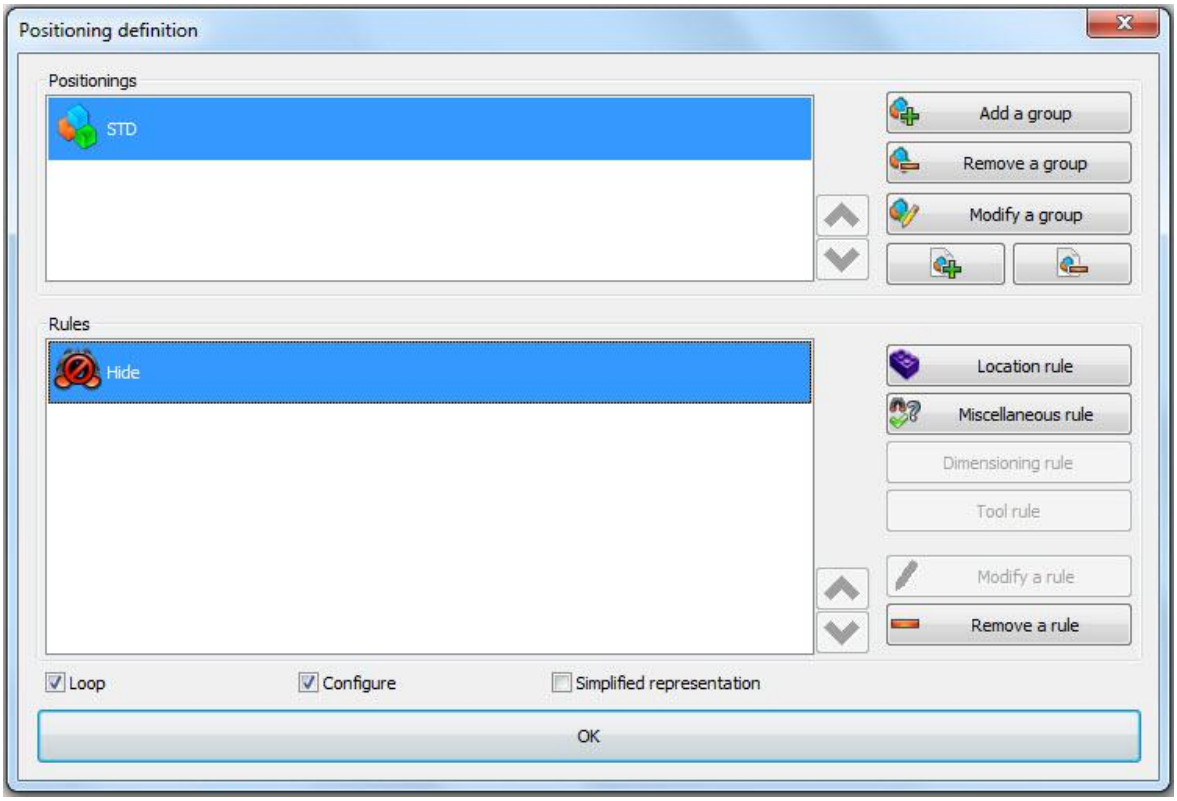
Using geometric drivers in TopSolid'Planner

The geometric drivers (especially the points) are now managed in TopSolid'Planner.
To be used, the component must contain geometric drivers.

*Example of a component **Triangular cover** controlled by 3 points.*

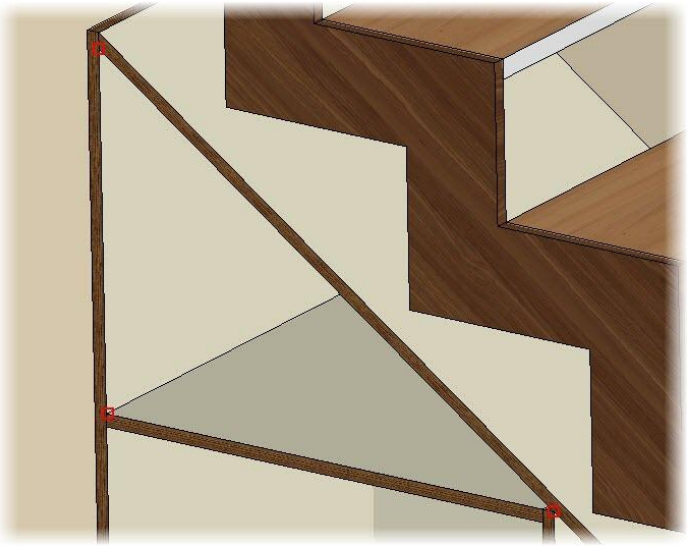


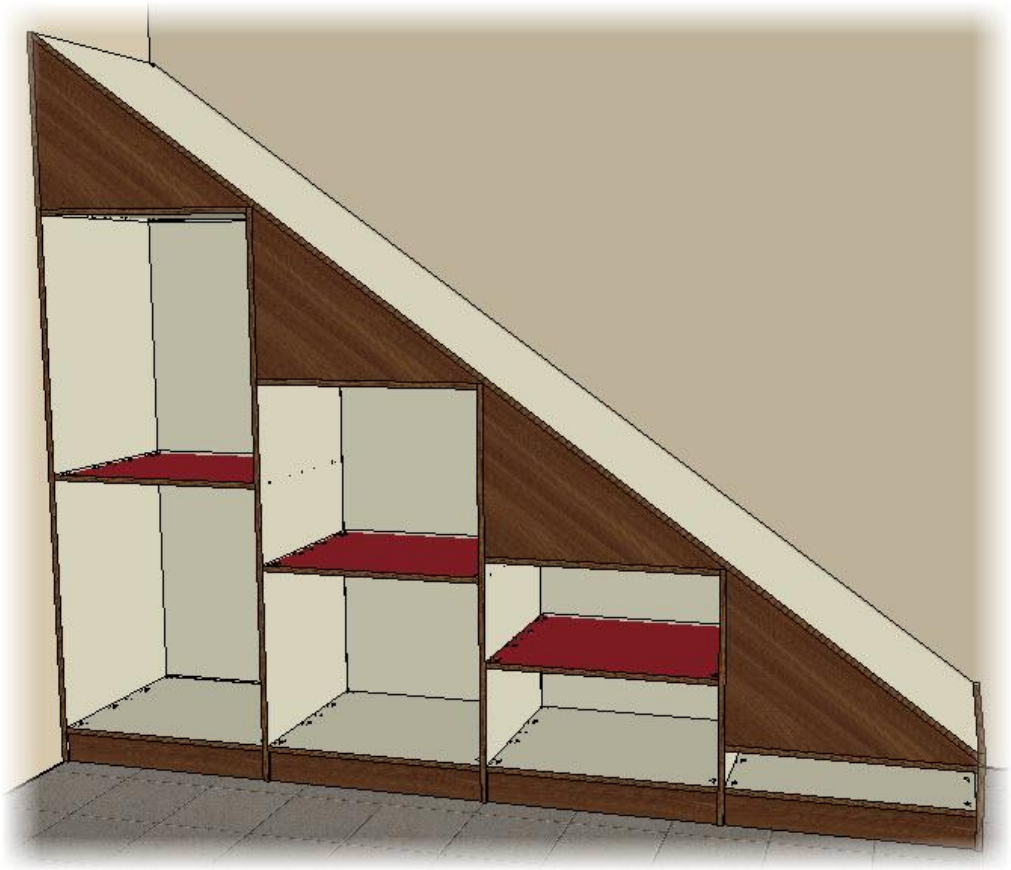
The component has to be saved normally in the Planner library.
Then, as with a driver block component, an empty positioning sequence has to be created.



The **Hide** rule allows you to hide the component during the geometrics drivers' selection.
The **Tool** rule can be added to create the component's tools.

When inserting the component in TopSolid'Planner, the geometric drivers are required and the component is included depending on the selected drivers.

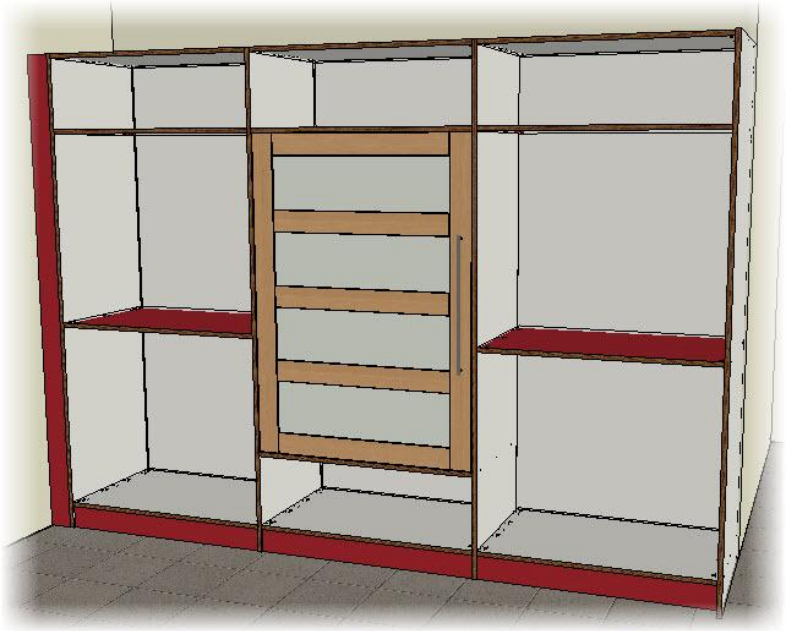




Detailed quote in TopSolid'Quote

It is now possible to show on a quote some subcomponents of an element on several quote lines. It allows you for example to display the handle type and the materials used in a component.

Example of this cupboard door.



N°	Product	Sketch	Q.	Unit	Unit € H.T.	Total € T.T.C.
5	Recessed door kit		1,0000		41,30	**41,30**

In the older versions, this door was displayed on one quote line.

5	Recessed door kit		1,0000		41,30	**41,30**
5.1	Cylindrical handle 512		1,0000		15,34	**15,34**
5.2	Glasse panel		1,0000		15,16	**15,16**
5.3	Recessed hinge		3,0000		3,60	**10,80**

Now it is possible to display the door's subcomponents on several quote lines.

Displaying the subcomponents

- Start by defining, in TopSolid'Quote, the *Door* product in the first level as Subassembly. To do this, edit the item and in the **Mode** field select **Subassembly**.

Note: By default, all products are in **Unitary** mode in order not to change the current TopSolid Quote's behavior.

The door and all its subcomponents are now displayed on several quote lines.

Hiding some items

Some items do not need to be shown in the quote (such as the hinges, for example).

- To not show them, edit the item and in the **Mode** field uncheck **Show item**.

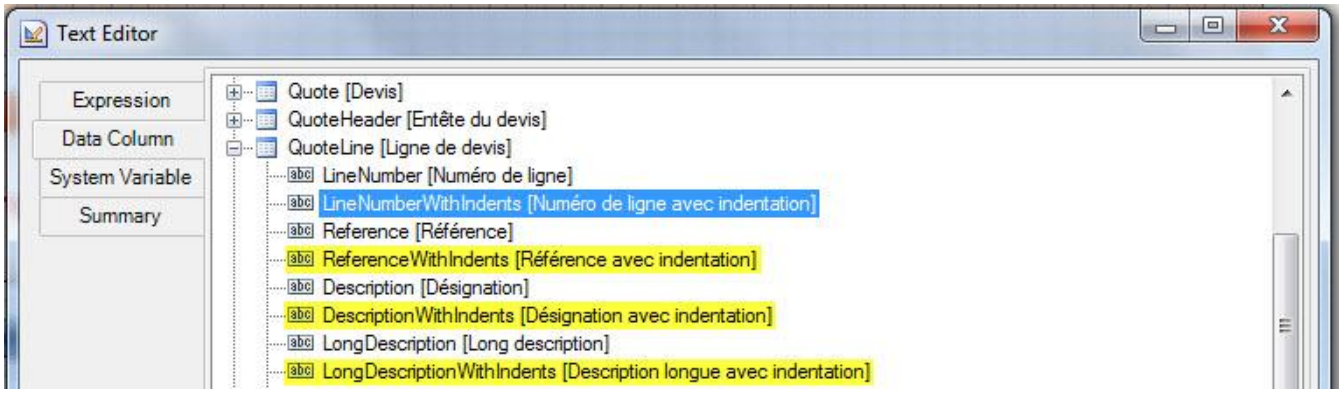
Shifting the data in the quote

To allow easier quote reading, it is possible to shift the quote's information (description, quantity, price…) according to its assembly level.

Here, it is the quote line's number which is shifted. At each new assembly sub-level, the quote line's number is shifted.

1
1.1
1.2
1.2.1

- To use this function, just modify the quote reporting model and edit the information case to shift. These new variables are in **Data Column | Quote line** and use the variable **with indents**.



Price displaying rule

TopSolid'Quote component's price displaying is now using a new rule in the **Quote detail** and **Quote preview** areas.

- Either it is only the superior component's price which is shown.
  If, when the component is unfolded, the subcomponent's prices sum is not equal to the component's price, only the component's price is displayed.

2	Recessed door kit		1	29,63	***29,63***
2.1	Cylindrical handle 256		1		
2.2	Glasse panel		1		

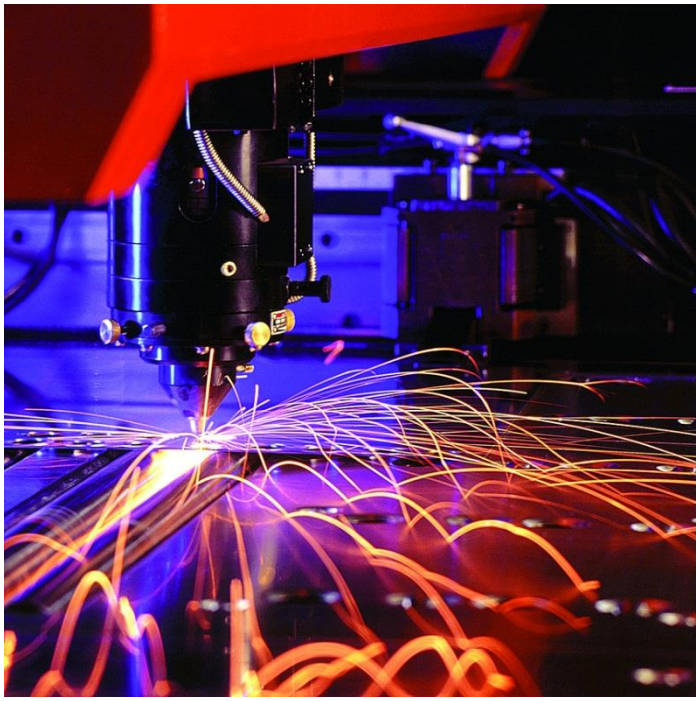
In the example above, the door kit is made up of a handle, a panel and three hinges which are not displayed. So, the displayed subcomponent's prices sum is different from the door kit price so only the door kit's price is displayed.

- Or only the subcomponent's prices are displayed.
  If, when the component is unfolded, the subcomponent's prices sum is equal to the component's price, only the subcomponent's prices are displayed.

2	Recessed door kit		1,0000		
2.1	Cylindrical handle 256		1,0000	7,67	***7,67***
2.2	Glasse panel		1,0000	11,16	***11,16***
2.3	Recessed hinge		3,0000	3,60	***10,80***

In this example, the displayed subcomponent's prices sum is equal to the component's price so only the subcomponent's prices are displayed.

TopSolid'SheetMetal 2013: What's New



Introduction

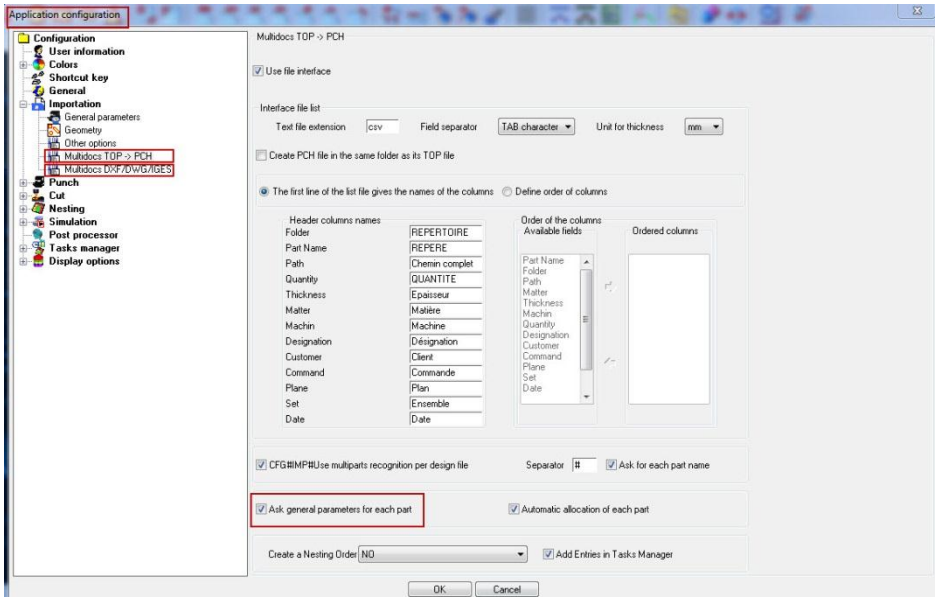
The 2013 version of TopSolid'SheetMetal includes many new features that will further improve your productivity. This user guide introduces the main new functions and the exercises accompanying this manual will help you to learn how to use them.

General

Import multidocs - Confirming general parameters

Purpose of the function

In this version, the user can access the general parameters dialog box when creating a part for the first time, when importing Multi-Docs TOP/PCH and/or Multi-Docs DXF/DWG/IGES.



Triggering the function

Tools | Options | Importation | Multi-Docs Top/Pch and/or **Multi-Docs DXF/DWG/IGES**.

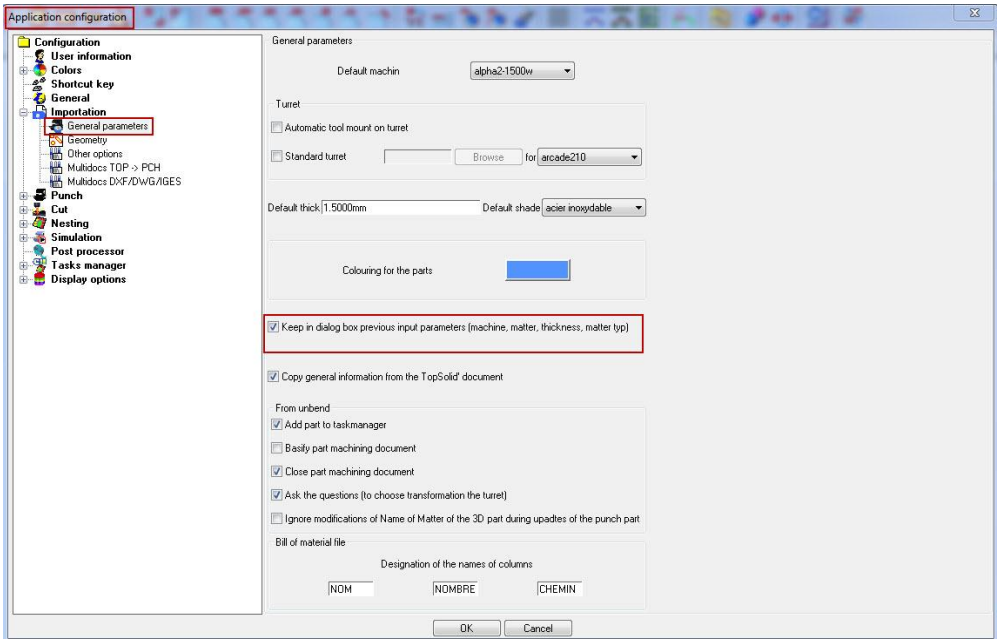
Keeping general parameters

Purpose of the function

Possibility of keeping the most recent values entered during the TopSolid session.

This concerns:

- The machine
- The matter (material)
- The matter type (material type)
- The thickness



Generating labels

Purpose of the function

Possibility of generating labels with set parameters.

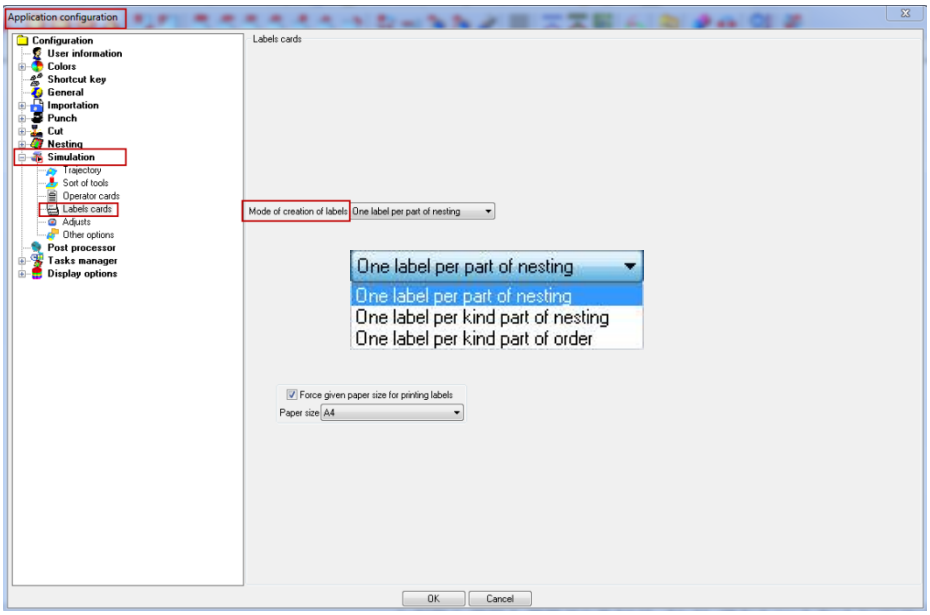
Triggering the function

Tools | Options | Simulation | Labels Cards

Select the desired mode to generate the labels amongst the available modes:

- One label per part of nesting
- One label per kind part of nesting
- One label per kind part of order

To use the labels, a label template model is required in the Punchdata directory.



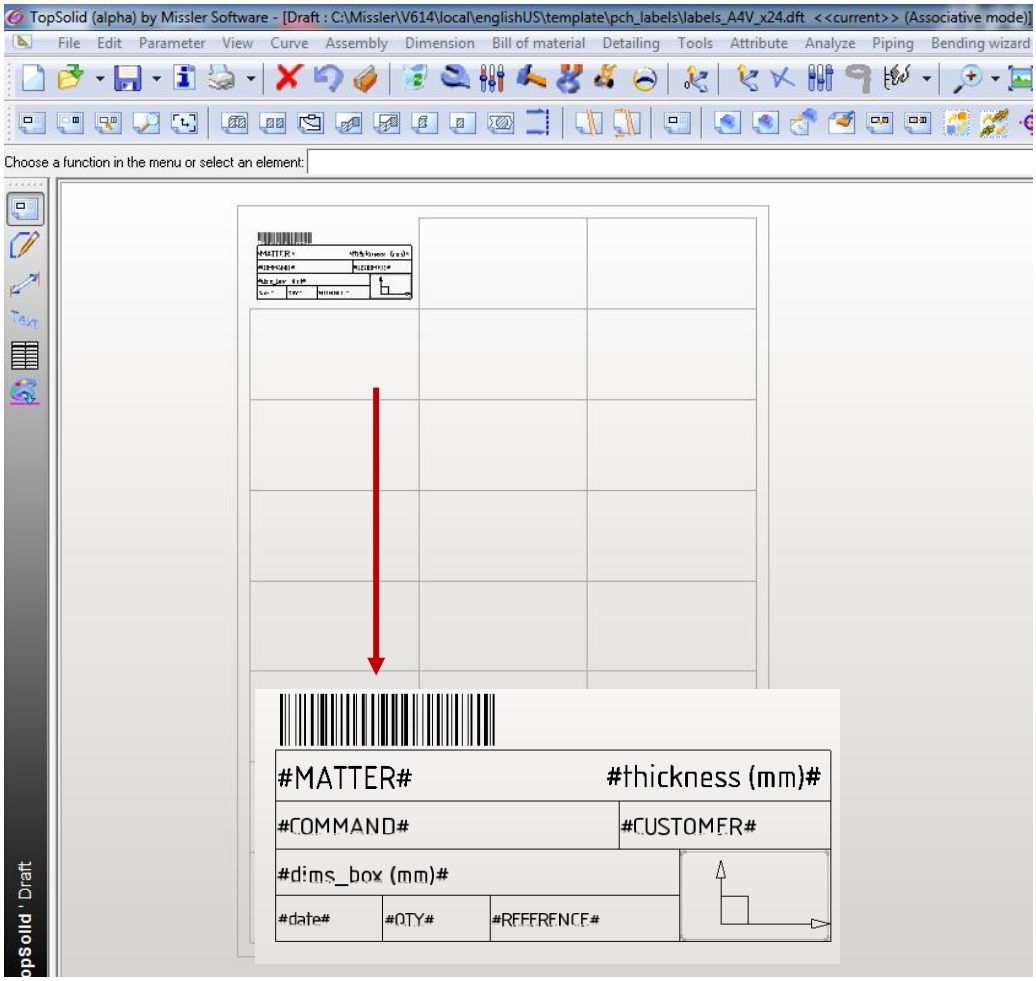
By default, the model shall be called *Labels.Dft*.
If you would like to create a model via machine, simply rename the file as follows:
Labels_Machine name.dft

Example: *Labels_bystronic.dft*

Models are available to users in the following directory:
\Missler\v614\local\french\template\pch_labels

Labels_A7V.dft	Model of a label in A7 vertical format
Labels_A4V_x8_1.dft	A4 template containing 8 labels
Labels_A4V_x8_2.dft	A4 template containing 8 labels (other model)
Labels_A4V_x16.dft	A4 template containing 16 labels
Labels_A4V_x24.dft	A4 template containing 24 labels

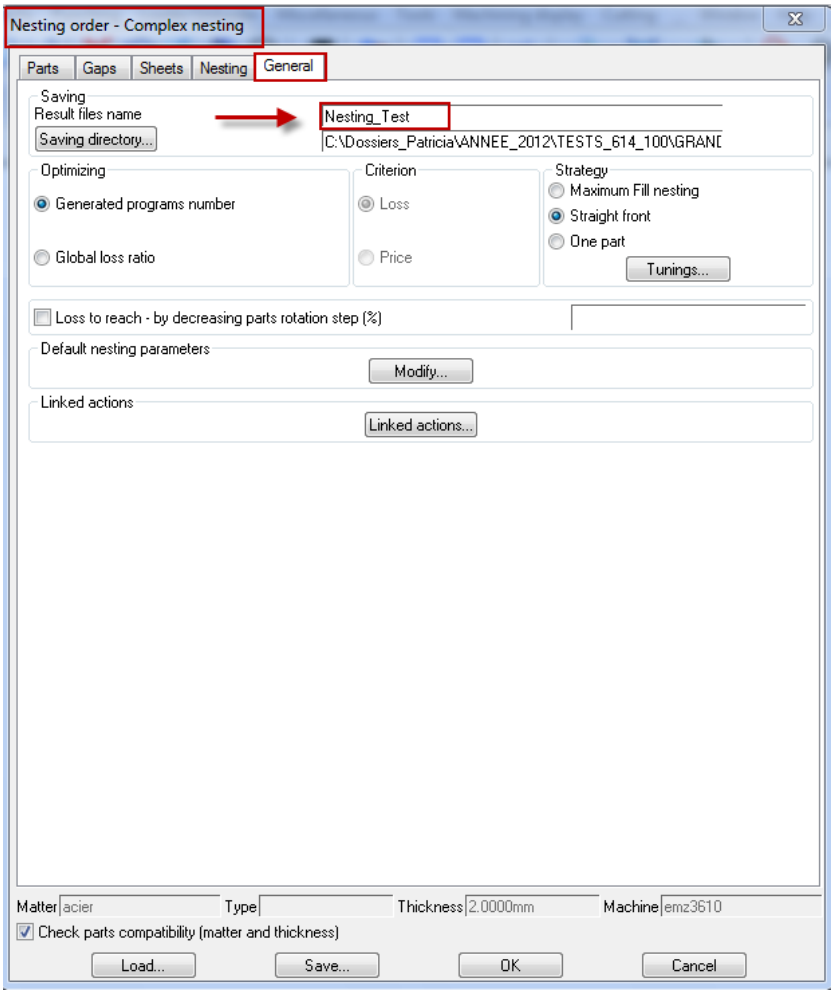
Example of templates:



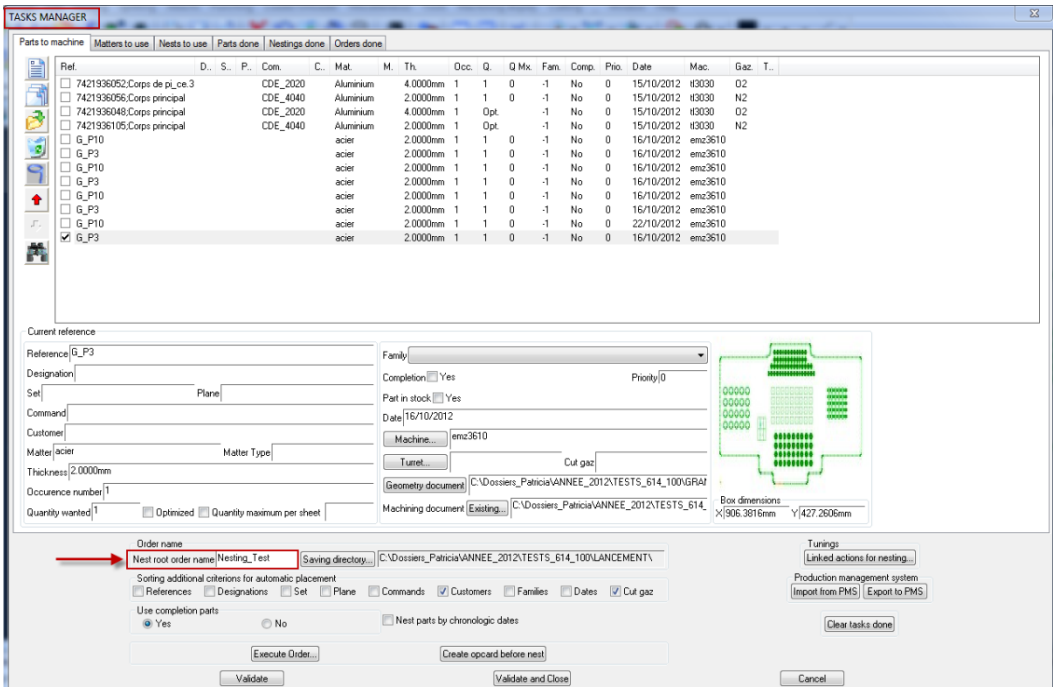
Extension of label files

Label files are Draft type files, with the extension **.Labs.dft**.

The label file created for a given nesting (mode 0 or 1) takes the same name as the nesting.



The label file created for a given order (mode 2) takes the same name as the order.



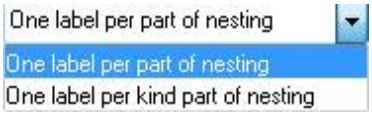
Manual creation function

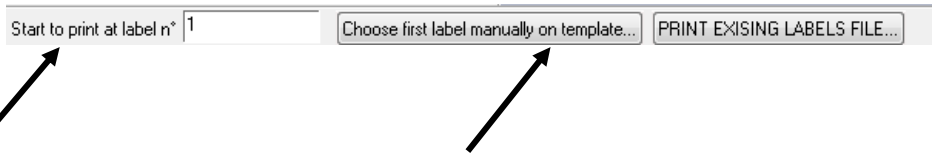
In the **Simulation** pop-up menu, a new function can be used to create and/or print a .dft file of labels.

Function name: **Create labels** 

Function procedure: Open a nesting document.

The combo box can be used to select the label creation mode.





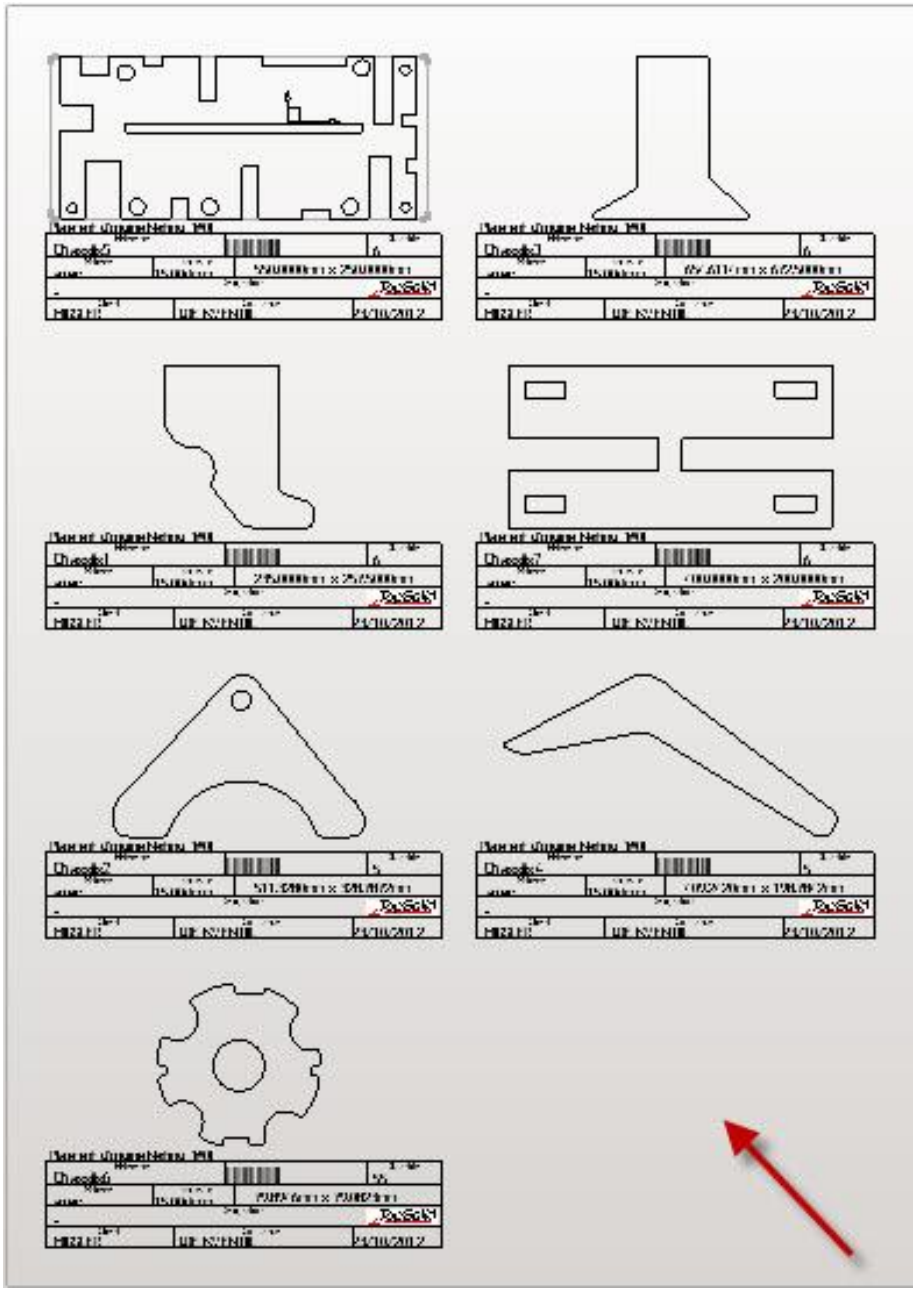
The following field can be used to indicate the number and the location of the first label. This field is for informational purposes and can be input using the keyboard.

This button can be used to directly select the first label manually on the template.

Example

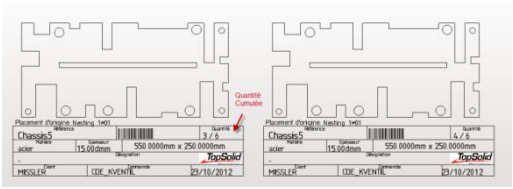
One label per kind part of nesting.

When the user generates another label file, the system automatically proposes starting with label no. 8.

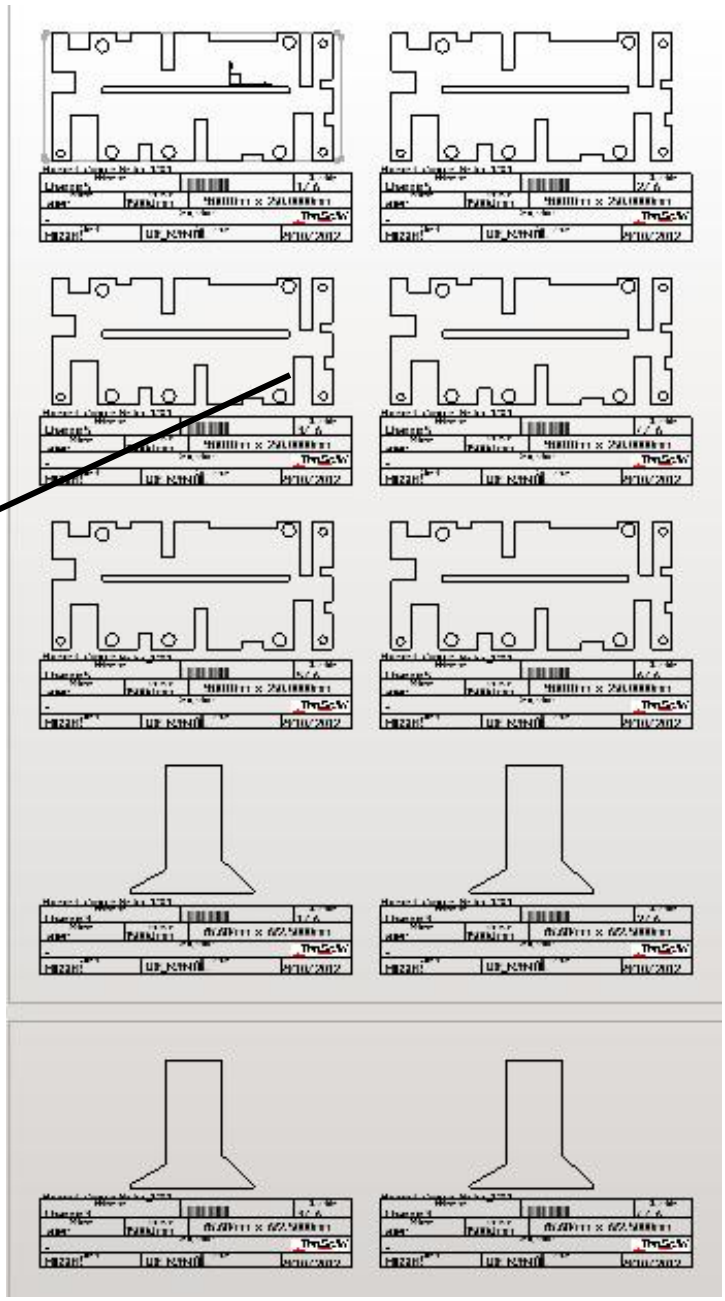


Example

One label per kind part of nesting.



Managing added quantities.

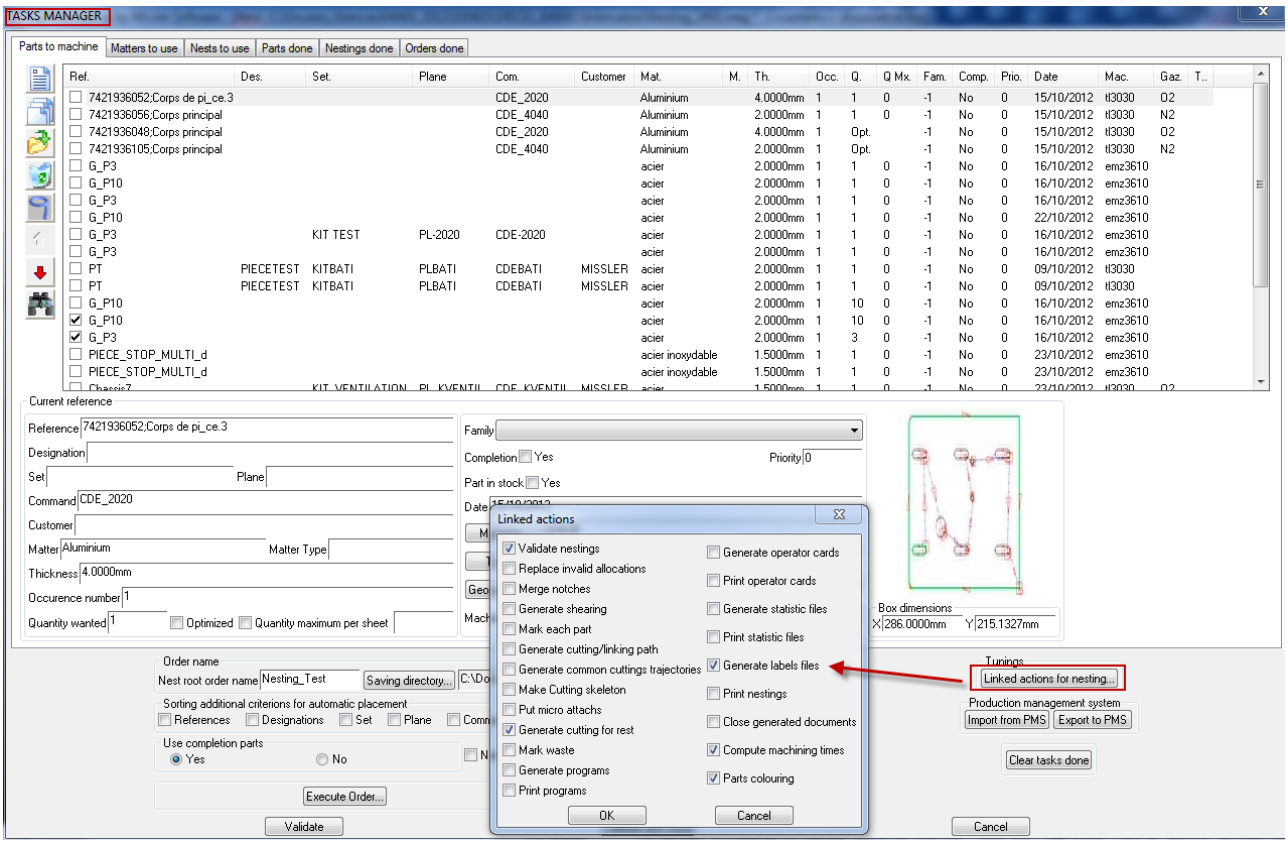


As soon as the label file has been created, the user can view and/or print it.

If the function is called up when the current document is blank, the function only proposes printing an existing label file.

Creating label files on the fly during nesting

A new action linked with the tasks manager can be used to create label files on the fly during nesting.

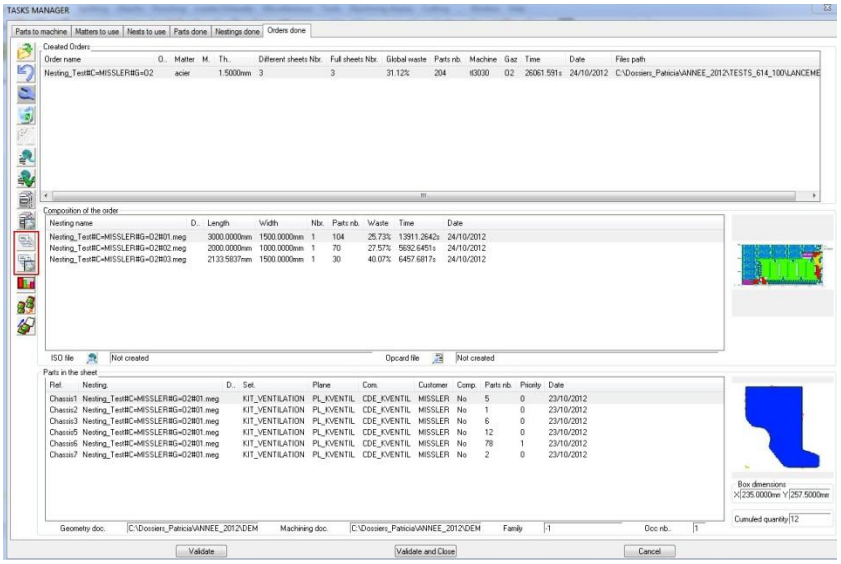


According to the creation mode required in **Tools | Options**:

- Either a single file containing all the labels for the various types of order parts will be created once all nestings have been made.
- Or a file containing all the nesting labels will be created (as the nestings progress).

Creation | Visualization | Print from the Tasks Manager.

Two new icons are available in the **Orders done** tab**.**



The first creation icon can also be used to recreate labels in the event that the nestings are subsequently modified (add/ move a part from one nesting to another, etc.).

The action depends on the focus position in the dialog box:

- If the focus is located in the **Orders done** zone, the function will create a file of labels that corresponds to **one label per kind part of order**.
- If the focus is located in the **Composition of the order** zone, the function will create as many label files as the number of selected order lines.

If the label files already exist, it will be possible to override a creation of new labels.
All the label files requested are displayed on the screen in tile format.

The **Print labels** function prints the label file(s) that correspond to the focus position in the dialog box. On the fly printing will only be initiated if all label files have been created previously.

Direct import TopSolid 7 - TopSolid V6

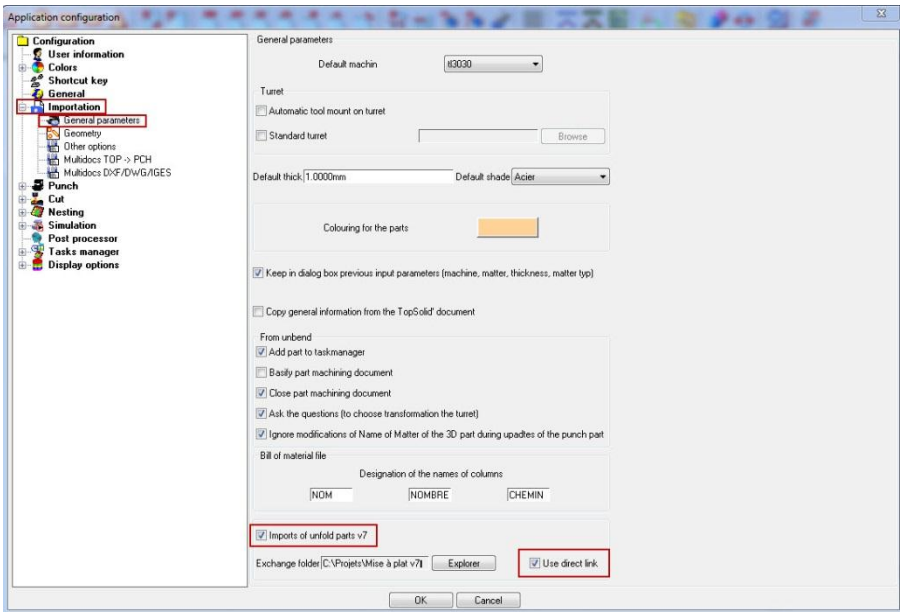
Purpose of the function

Direct link between TopSolid V7 and TopSolid v6.

Triggering the function

Tools | Options | Importation | General parameters

Modifying these parameters requires restarting TopSolid.



SMI link

The **SMI** link has been improved. In previous versions and during document upgrades, V7 was intended to export the TopSMI document with the same name and in the same directory as for the initial import.
In version 6.14 and on, using the **exchange directory**, TopSolid V6 finds the TopSMI document in order to update the machining part, even if the name of the exchange document is not the same as for the initial import.

Direct link

When TopSolid V7.7 is installed on the same PC as TopSolid V6.14, the direct link can be used without having to navigate through TopSMI unfold parts.

Box available in **Tools | Options**.

Method

In version 7, unbendings must be checked in.

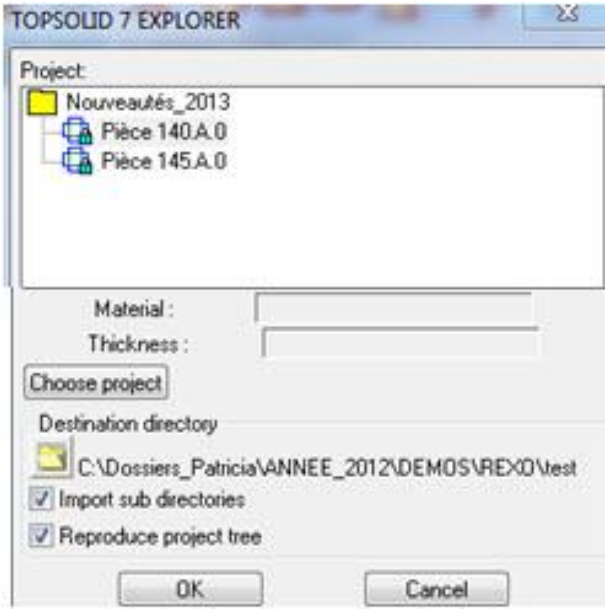
- In a TopSolid'SheetMetal **Part** document, use the **New part** function  .
- Select either the **V7 Explorer** button or the **Current V7 Project** button.

V7 Explorer

In the event that TopSolid V7 is not loaded, it will be loaded in memory and the project explorer can be used to select the part to be imported.

Only files that contain unbend parts ready to be imported in V6 appear in TopSolid7 explorer.

- To switch projects, use the **Choose Project** button.
- Continue the import by selecting the machine.



Current V7 project

Current V7 Project explorer only displays the file that contains a current unbend part in V7. This file must have been checked in beforehand in V7 to be used in CAM V6.

- Continue the import by selecting the machine.

Update 

The **Green/Red** light can be used to indicate, in real time, if the V7 part has undergone changes.
If the light turns **Red**, click on the light to update the PCH part using the V7 unbending.

The light turns green.

The  icon indicates a connection to TopSolid V7 (= TopSolid V7 activated).

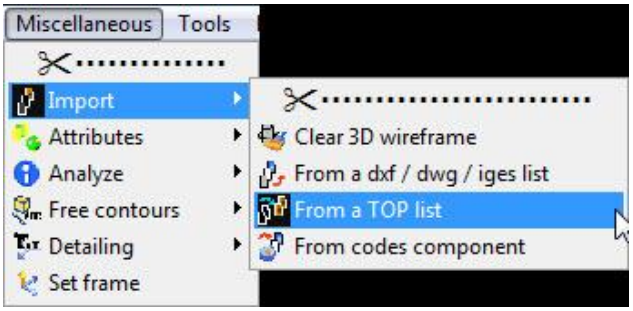
Working offline is possible (= TopSolid V7 deactived) by clicking on the  icon.

The update status is temporarily unknown  .

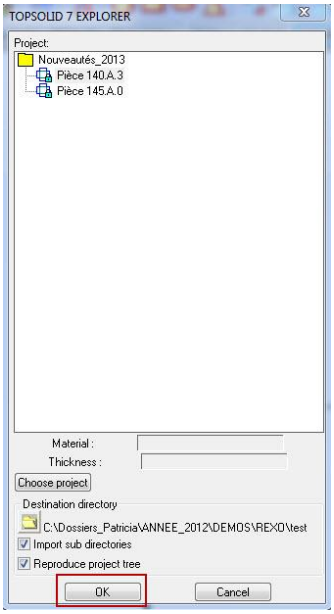
Offline mode makes the import options unavailable from V7.

Importing multi-docs (using the direct link)

From the **Miscellaneous | Import | Multi-documents | Top-→Pch** menu, use the **V7 Explorer** button.



The project explorer can be used to select all the V7 unbendings in a file and to view all the unbendings for an entire project.

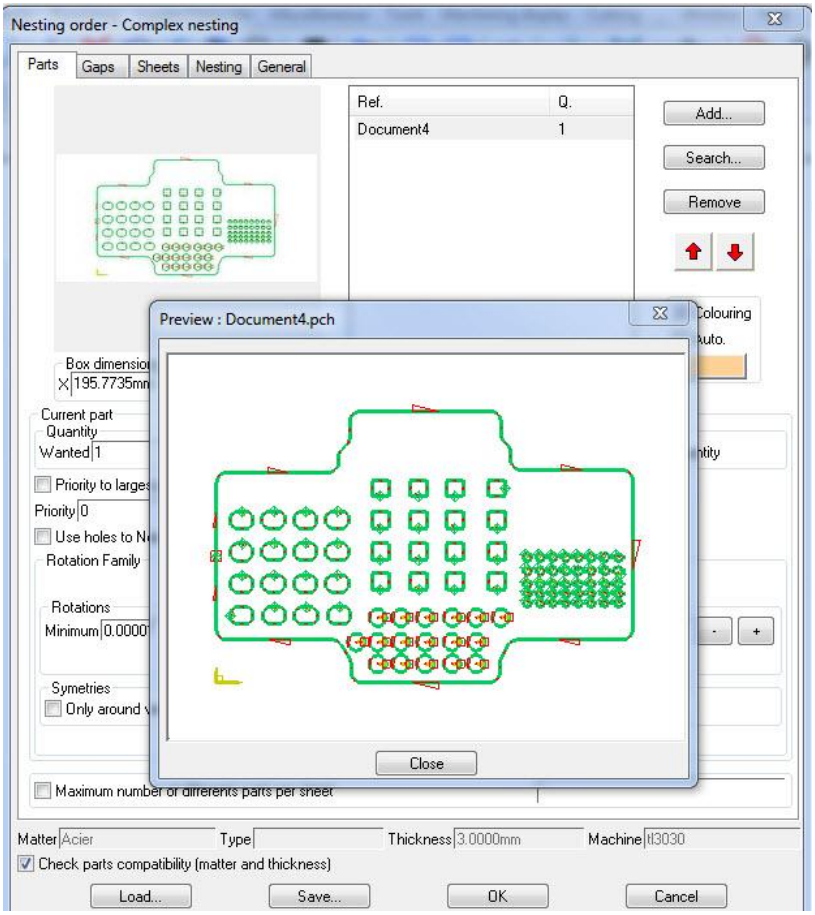


User interface

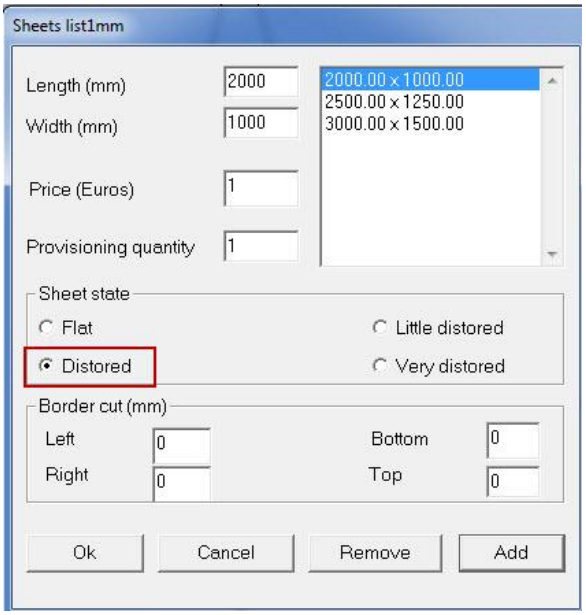
Previews

In the **Nesting Order** dialog boxes as well as in the **Tasks Manager** dialog boxes, there is the option of clicking on the Part or Nesting preview, which temporarily enlarges the view in a dialog box.

The scroll wheel can be used to zoom.



Sheet flatness



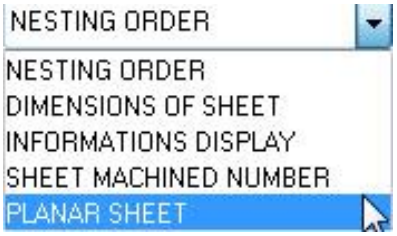
Modifying sheet flatness

When defining the sheet formats in materials management, it is possible to indicate a given state of flatness for each format (associated with a material and a thickness).

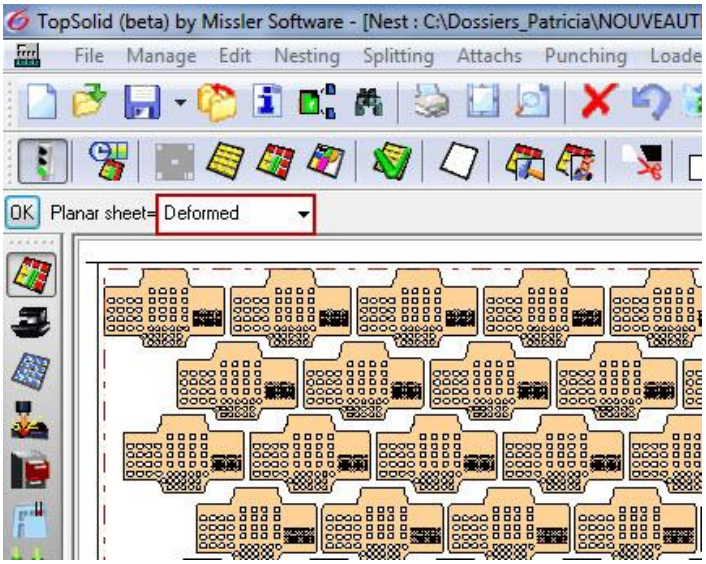
This state can be reworked once nesting is done.

Edit function

- Use the wrench to select the format on the screen.
- Expand the combo box, select **Sheet state**.
- Expand the combo box, select the new state and click **OK**.



When editing a nesting and if the format size changes, the flatness state is updated according to the new dimensions.
Likewise when a change has been performed using the wrench with the **Format dimensions** option.



For non-standard formats that are not defined in format management, a default state shall be entered in **Tools | Options**.

SheetMaster

Dynamic unloading adjustment

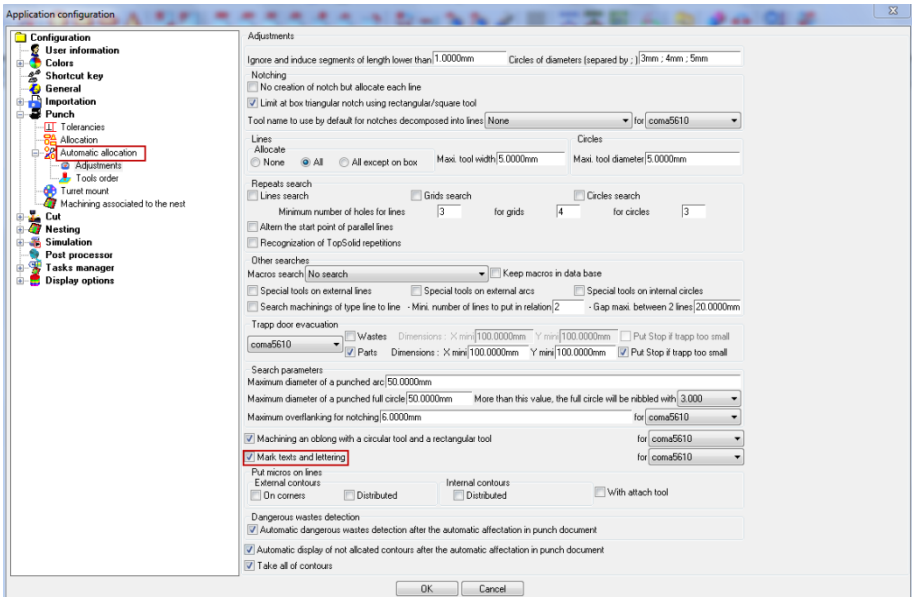
When unloading a part or a waste with SheetMaster, it is possible to provide a delta value position gap (in relation to the punch) along the X axis (or along the X and Y axes depending on the machine).

Punch - TopSolid'Punch

Punch using the marking tool

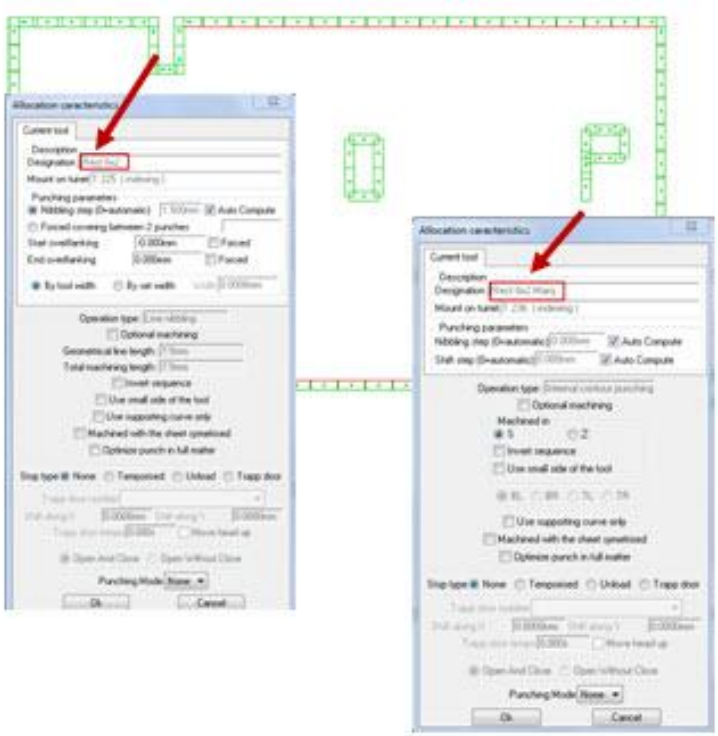
If using the font *lettering-Punch-RC.top*, the text is broken down into small rectangles with dimensions provided according to the required text height.

If using automatic allocation, in **Tools | Options** it is possible to mark texts and the lettering and to use the rectangle tool in the **Marking** category.
Therefore, this tool could be used on other geometries of the part that do not depend on the **Lettering** function.



In version V6.14, even if there are two tools with the exact same dimensions: one classified **Normal** and the other classified **Marking**, automatic machining will exclusively select **marking** for lettering, and the **normal** type tool for punched geometries.

The font characteristics must match the tool dimensions exactly.



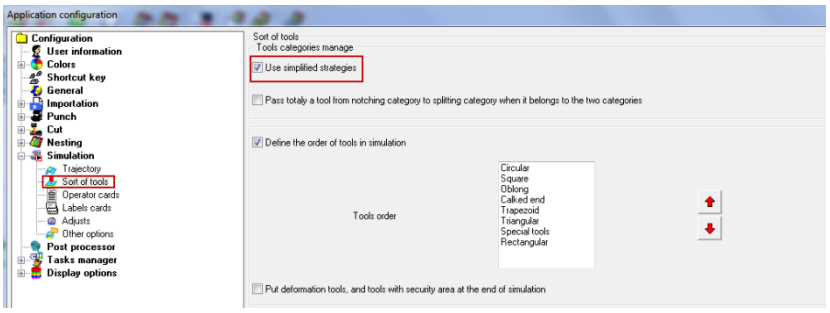
Strategy Changes

Improved preservation of tool order

Significant improvements have been made to better preserve changes in tool order carried out in the **Order of operations** dialog box during updates and when deleting nesting parts.
A new check box in the **Options** tab of the strategies dialog box can be used locally to return to the default order without completely resetting the strategies.

Also, the system now alerts when the previously made changes cannot be kept.
When a nesting changes and there are more zones than before (offsets), it is possible to apply the operation order changes carried out in zone 1 to the new zones.

Simplified strategies

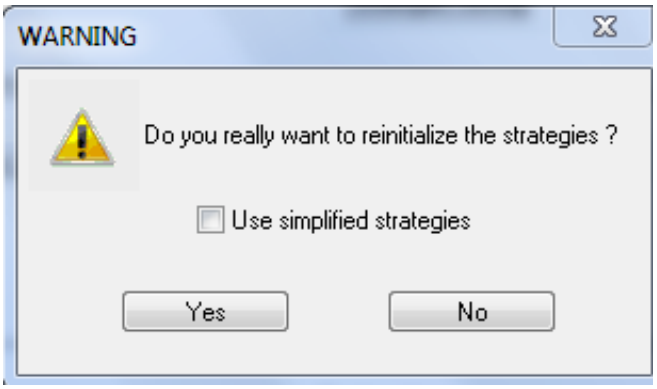


A new option makes it possible to use simplified strategies.

By default, this mode only uses the **Internal cuttings** and **Punchings and Evacuations** categories. This makes it possible, among other things, to limit the tool changes when a single tool is used on both internal and external contours.

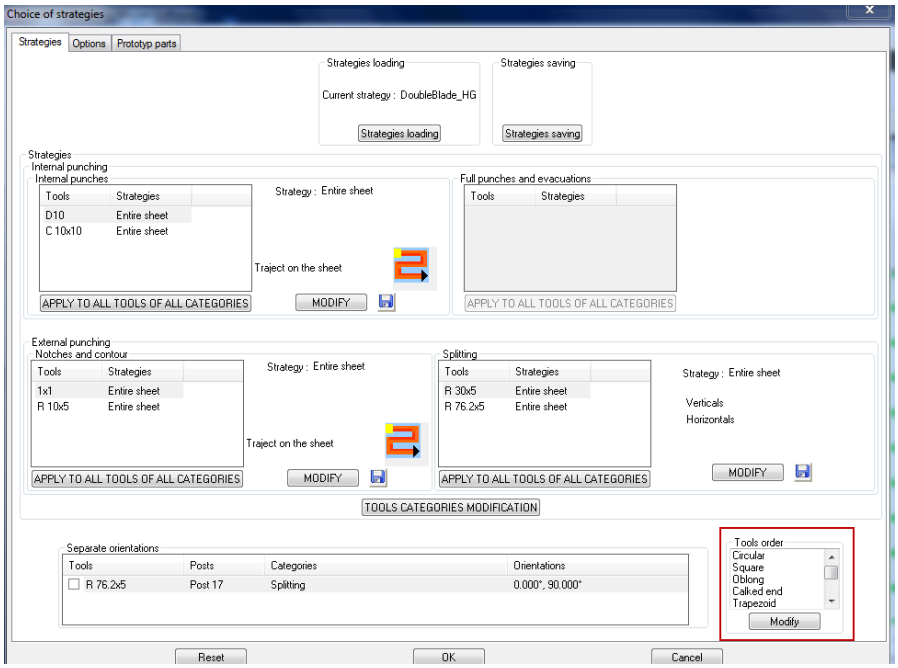
It is possible to move a tool to the **Notching and contours** category or to the **Splitting** category at any time using the **CHANGE OF CATEGORIES** button in the dialog box **Choice of strategies**.

When you reset the strategies (using the **Reinitialization** button in the dialog box) or when you reset the simulation (using the player button), it may or may not be possible to use simplified strategies.



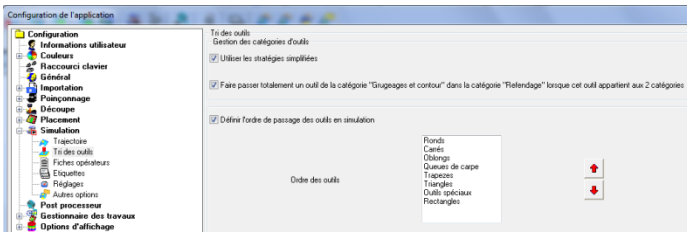
Order of tool shapes

The order of tools according to "shape" is now memorized in each nesting. It appears in the choice of strategies dialog box.



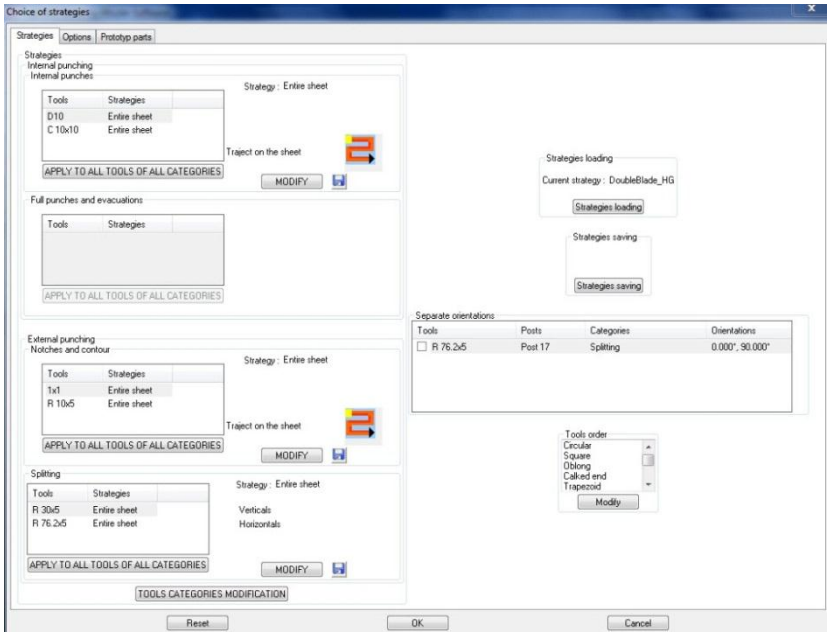
To change the order of the shapes, click the **Modify** button.

Reminder: Tools classified as **splitters** do not take this order into account and will always be placed at the end of the list.
The default order at the creation of the first simulation is always defined in **Tools | Options** in the **Sort of tools** section.

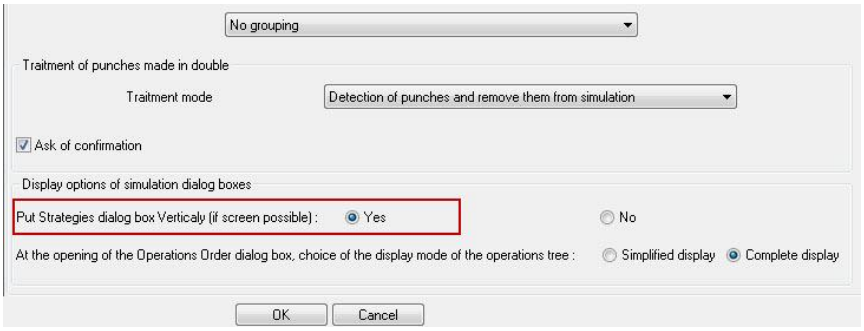


Change in appearance of the choice of strategies dialog box

In order to improve the overall legibility of the tool order, categories can be positioned vertically in the choice of strategies dialog box.

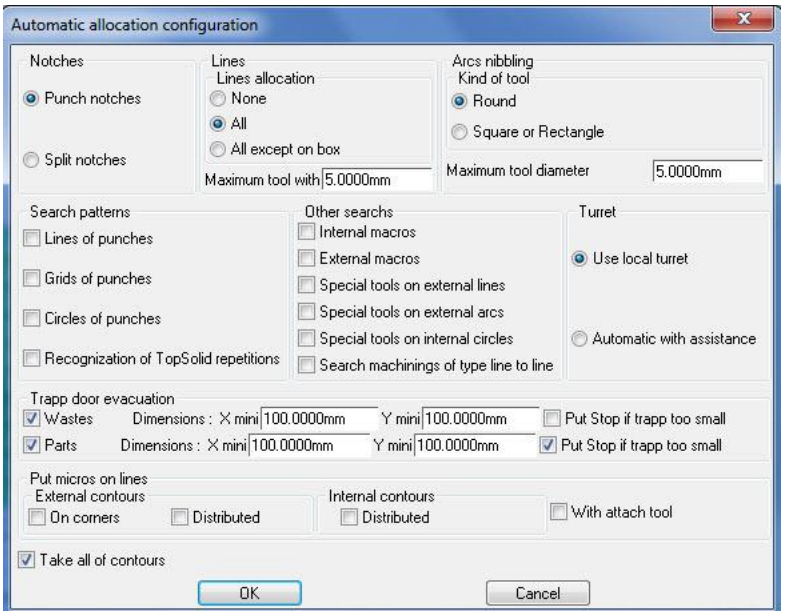


The configuration is carried out in **Tools | Options** in the **Simulation Sort of Tools** section.

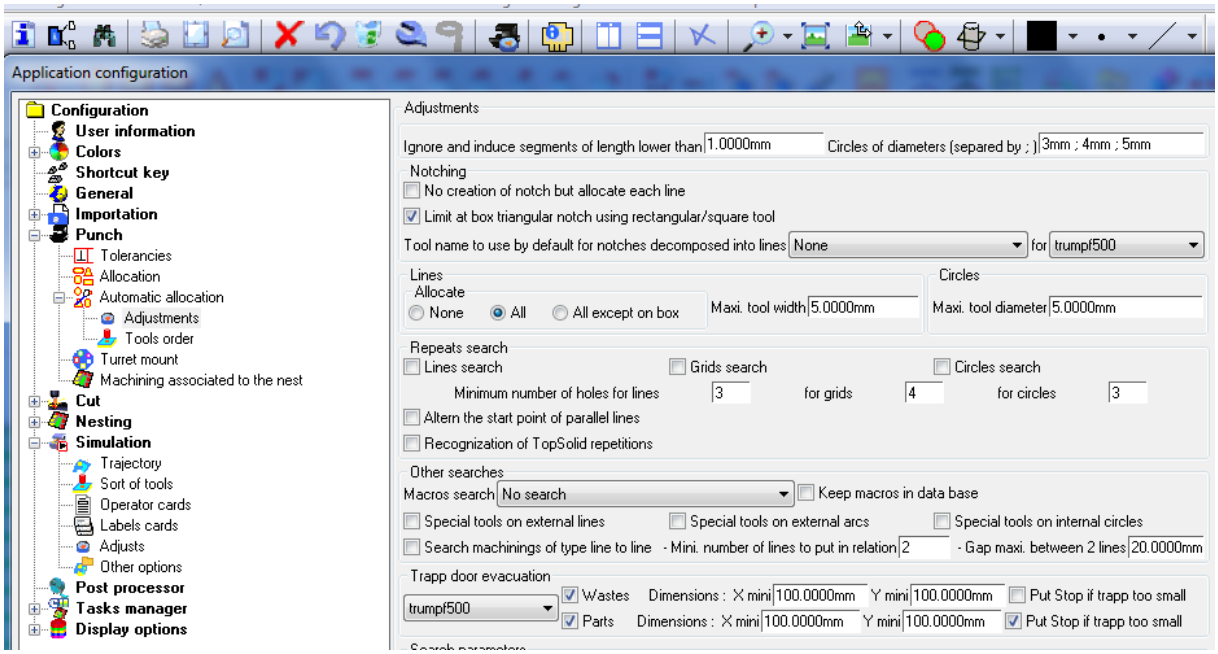


Trapp too small – Put a STOP

At the time of automatic allocation during punching, possibility of automatically evacuating the part or the wastes via the trapp using the dimensions provided. If the waste or the part is too big for the trapp, no evacuation was programmed. Version V6.14 makes it possible to position a Stop automatically in this case.

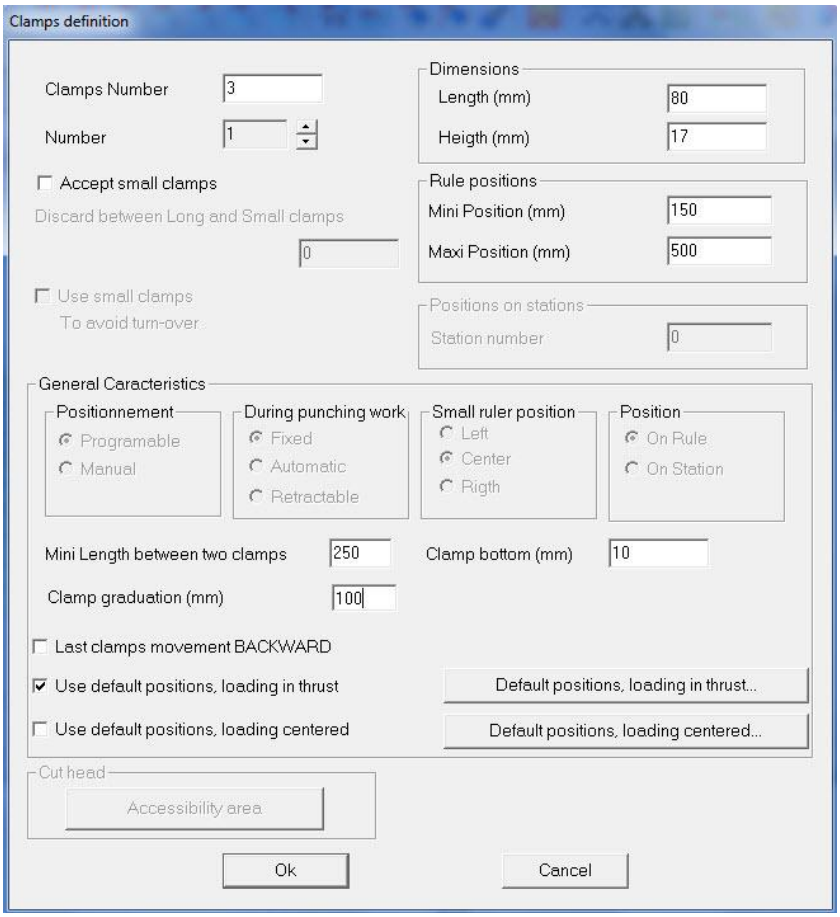


Configuration in **Tools | Options.**



Graduated clamps

A new machine parameters option can be used to define the clamp graduations. By default the value is set to 0 which means that the clamps can be positioned freely on the ruler.



When a non-zero value is indicated, and you have not **used the default clamp positions**, the system calculates the positions by balancing the clamps on the sheet and rounding to the nearest graduation.

When you move the dynamic clamp and when you click on the final position, the clamp is reset to the nearest graduation.

Reminder: If the graduation value is set to 0, the automatic automatic positioning of the clamps places them by rounding to the nearest mm. If the graduation value is set to 0, the dynamic manual move positions the value at the picked position, which is not rounded.

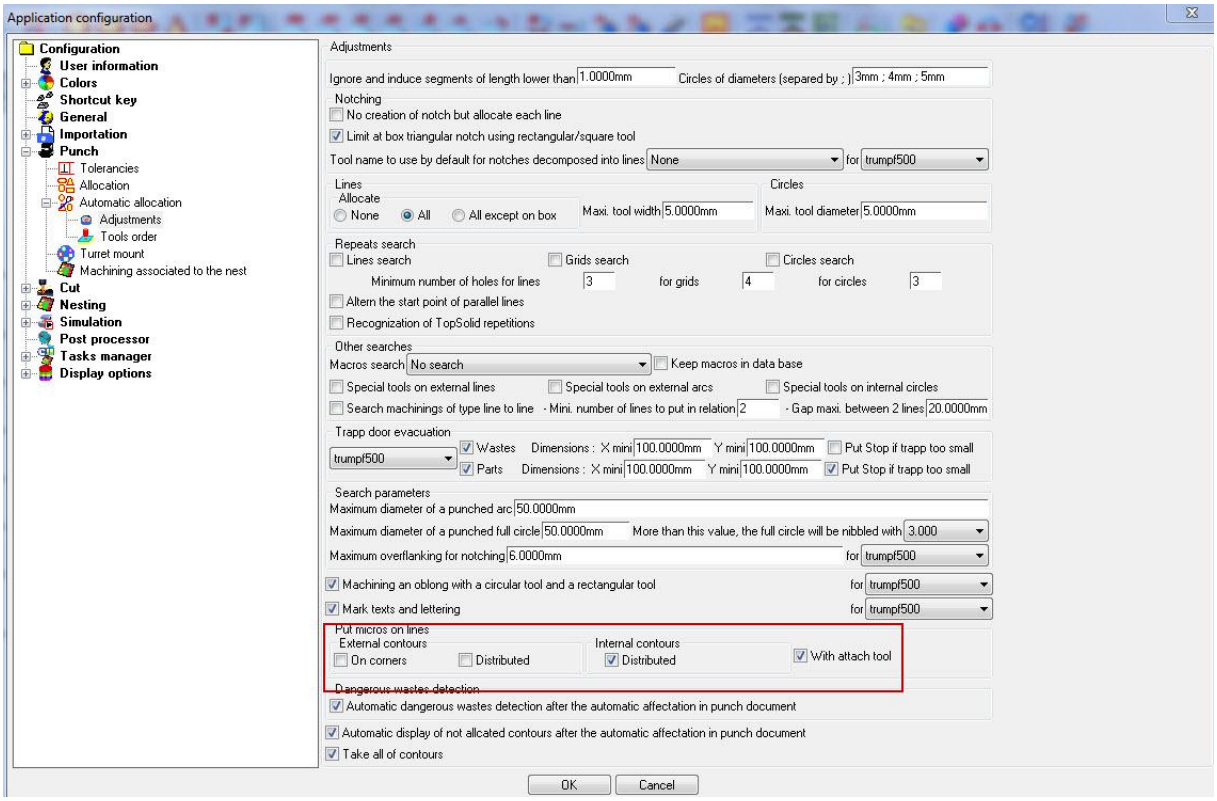
Turret-Color for unused tools

When you modify the turret of a part or nesting, the stations of the tools mounted but not used are indicated in orange.

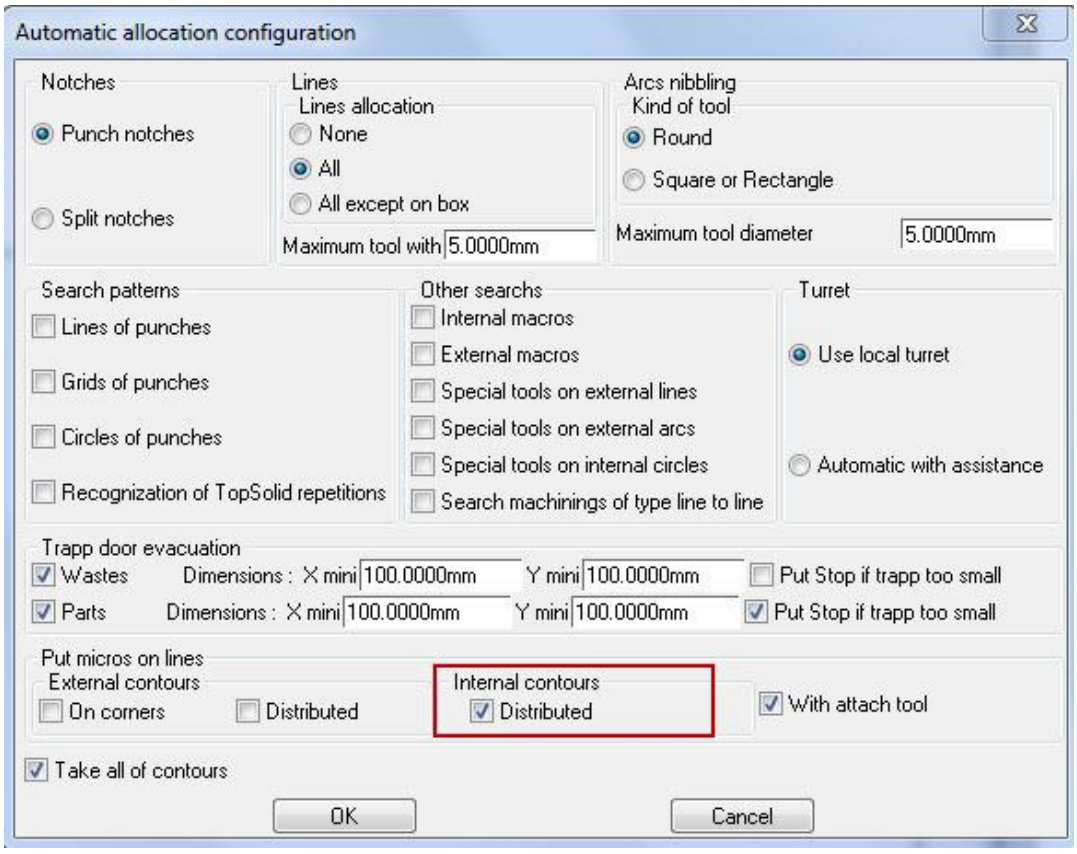
Automatic punching - Micro joints

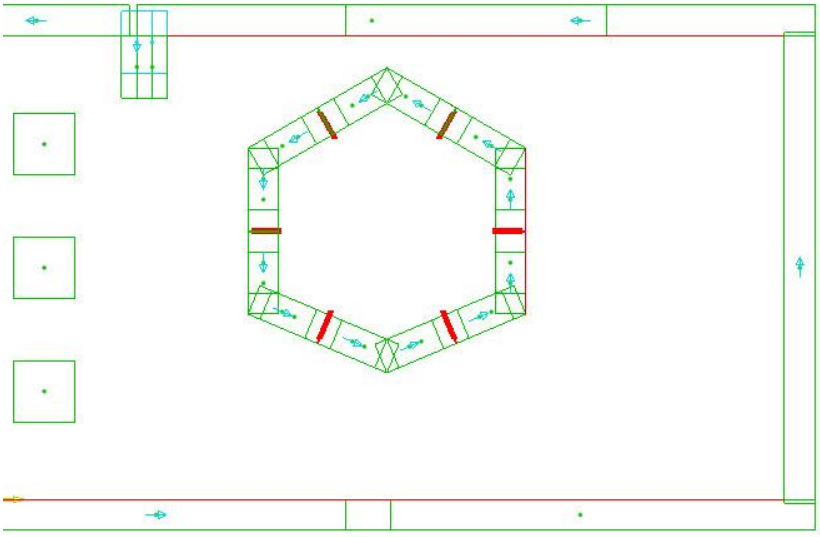
When automatic machining during punching, it is possible to position the micro joints distributed on the internal contour lines.

Settings in **Tools | Options**.

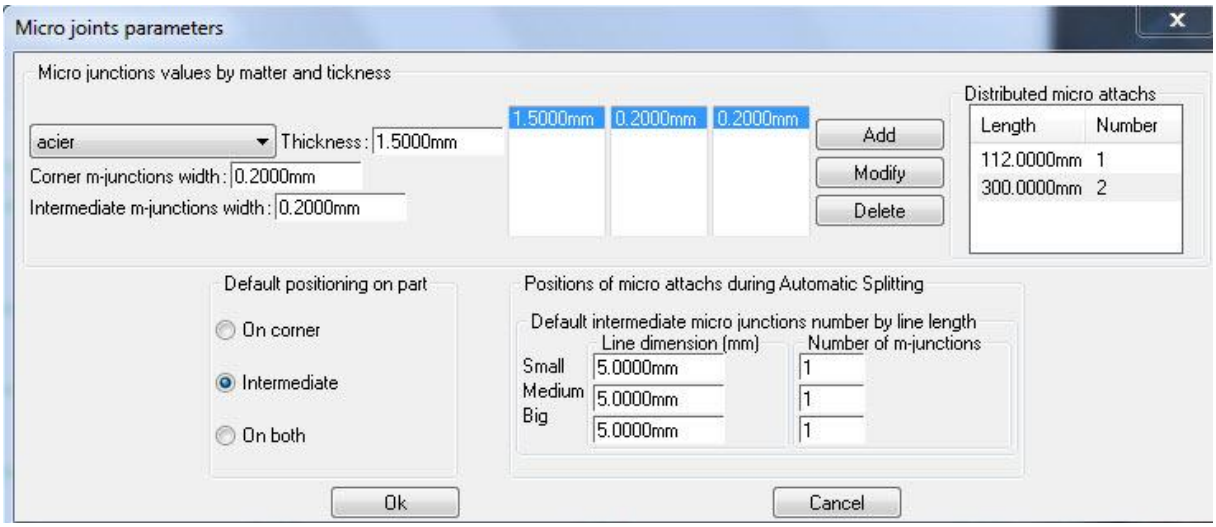


During automatic machining, this option is available locally.





Settings for the number of micro joints according to the length of the line is carried out in micro joint management.



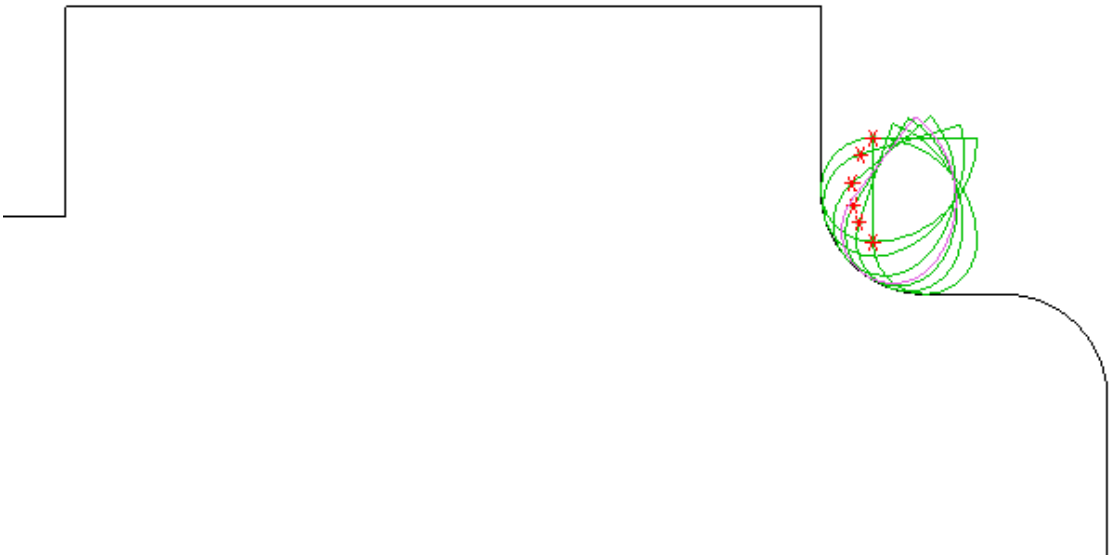
Micro joints are only distributed if the item does not already have micro joints at the start of automatic machining.

Special tools on points along profile

The function can be used to place a tool on along profile, along a profile that has been expanded for the use of special tools.

Option: **Along Segment**

- After having selected the tool, click geometry and then select the tool segment that will slide on the geometry.
- Click the final position.



Option: **Extremity Segment (with collision control)**

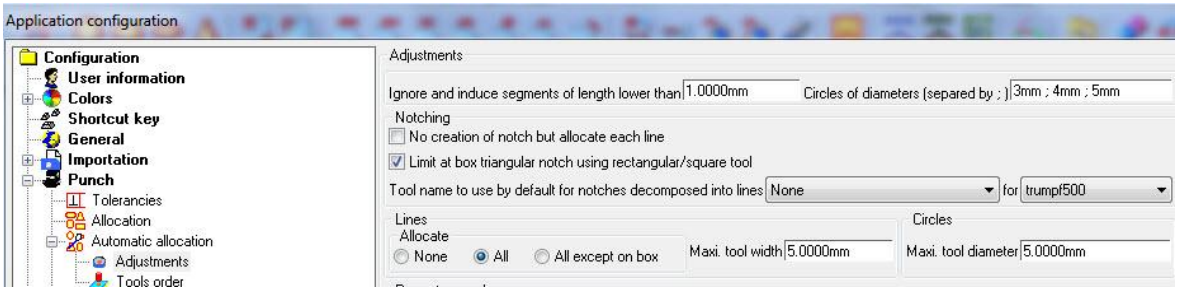
- After having selected the tool, click on the geometry nearest the desired extremity and then select the tool segment that will be used.
- Then click **OK** to confirm.

In the event that the geometry has been modified, the positioning remains associative.

Induced machining (automatic punching)

During automatic allocation during punching, it is possible to ignore segments whose length is less than a certain size by setting them induced.

Settings in **Tools | Options.**



If the value is set to 0, all segments will be machined.

Likewise, it is possible to define the values of the circle diameters that will be ignored and set induced (separate each value using the ";" character)

Simulate the tool change before repositioning (with head) if deformation

On TRUMPF machines that do not have the jack tightening option, you are required to tighten the sheet using the head when repositioning. However, if the tool in the head is mounted on the multitool or even if this tool is a special, not "normal" tool (e.g. deformation), it is necessary to mount an adequate tool in the head before repositioning.

In this case, the machining simulation is now compliant.

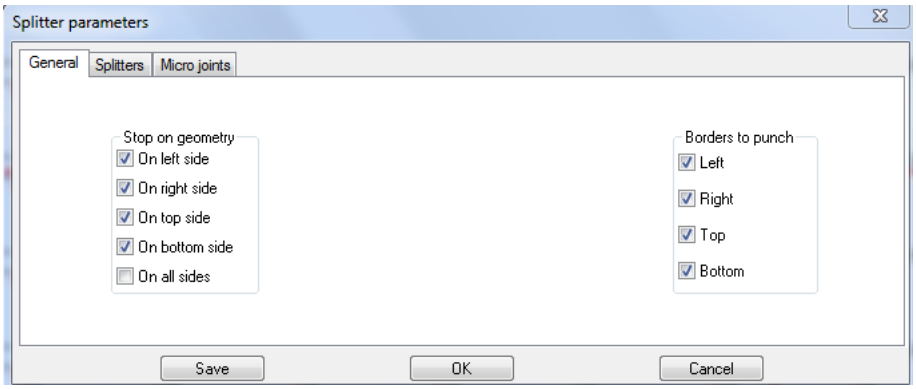
Automatic Splitting - Micro joints

The function can be used to create automatic splitting that now makes it possible to take into acount micro joints already present on parts.

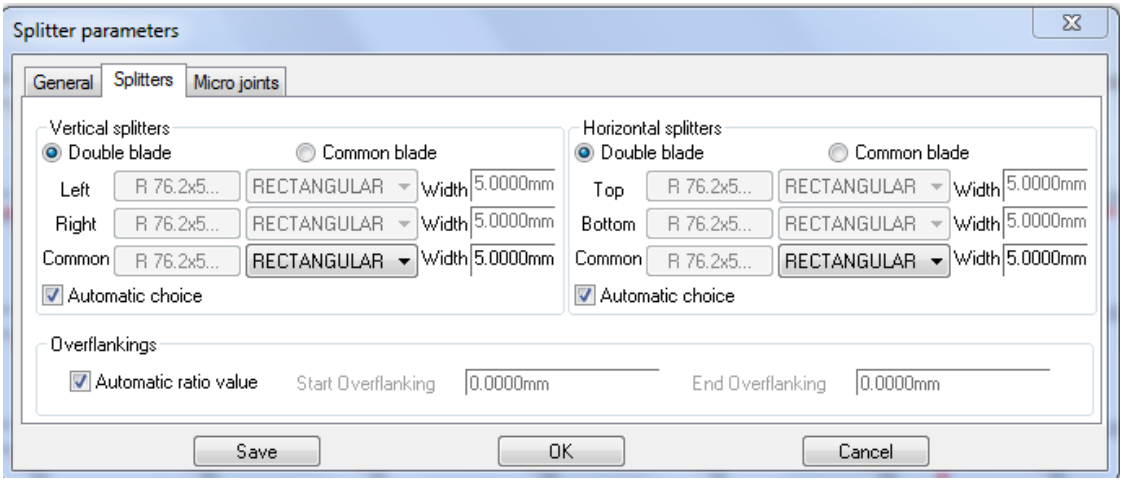
New **Without evacuation** option from the function:

When launching the **automatic splitting** function, a new button **WITHOUT EVACUATION** appears at the end of the line.

This option opens the dialog box that can only be used to select the edges to be punched and the tools.



No evacuation or micro joint will be added.



Possibility of using standard functions to locally add, modify and delete micro joints on the splitting lines created. You can also add micro joints manually, or distributed on item. Picking can be carried out on the splitting line created or even directly on the part geometry.

Restriction: Distribution options on contour, positioning in corners and searching for part corners are not possible if the automatic splitting lines were created previously.

The **machining** or **part** propagation function can be used to transfer micro joints of the splitting lines of a part to the splitting lines of other similar parts of the nesting.

If the micro joints were positioned on the parts prior to creating the automatic splitting with the **without evacuation** option, these micro joints will be taken into account.

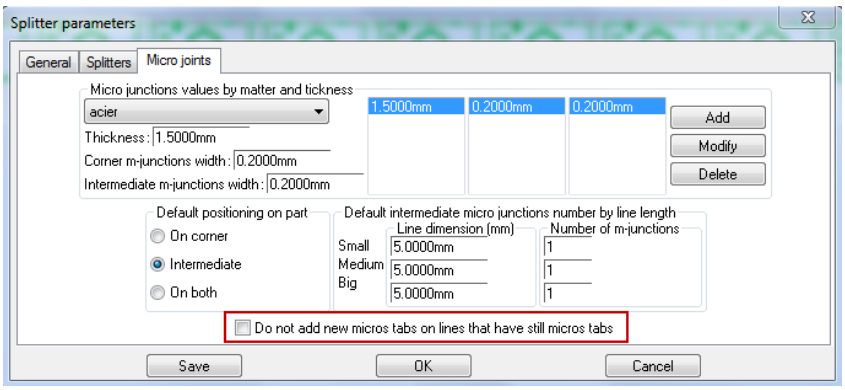
It will be possible to modify the size locally, to add, delete and propagate these micro joints.

Micro joints option of the function:

This option changes the behavior if micro joints are already present on the parts on which you wish to create automatic splitting.

By default, the micro joints present on the parts are going to be added to the new micro joints requested (in corner and/or intermediary).

A check box can be used to add new micro joints on the lines that already have them.



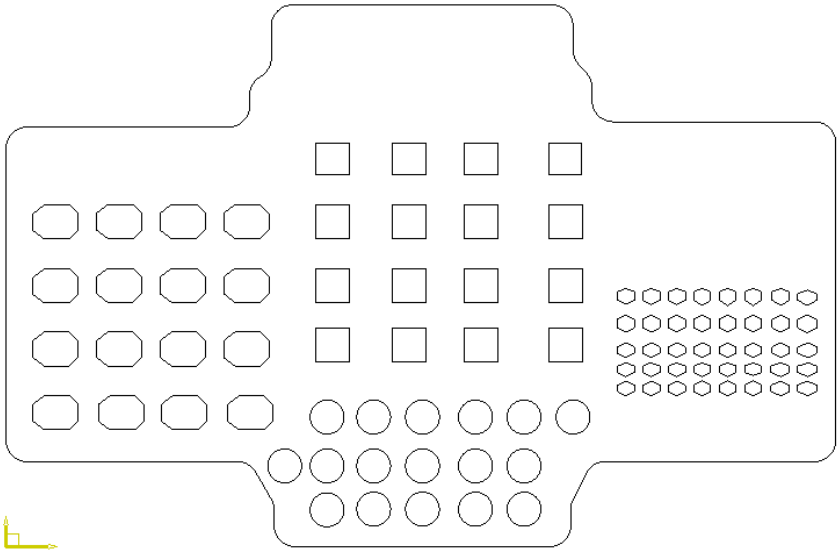
Cut - TopSolid'Cut

Fly cut path

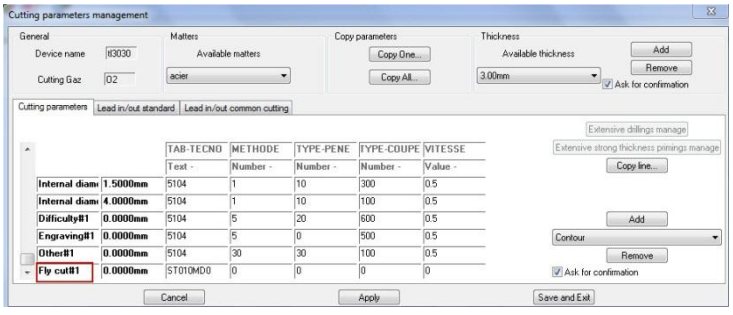
Fly cut paths can be performed on certain next-generation TRUMPF machines in the thin sheet and on even cuts that are close together.

E.g. grids of rectangular holes.

These cuts can be made without lead in, the cut tool path does not operate on the full contour but passes from one contour to another.



It is essential to have defined the **FlyCut** cutting parameter beforehand.

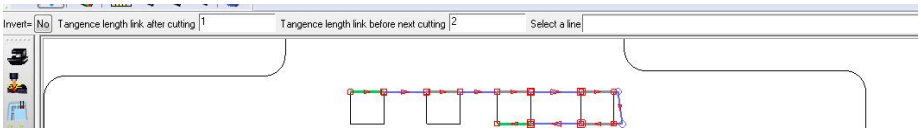


Access to the **Cutting** menu is available in the Part document.

Two modes are available:

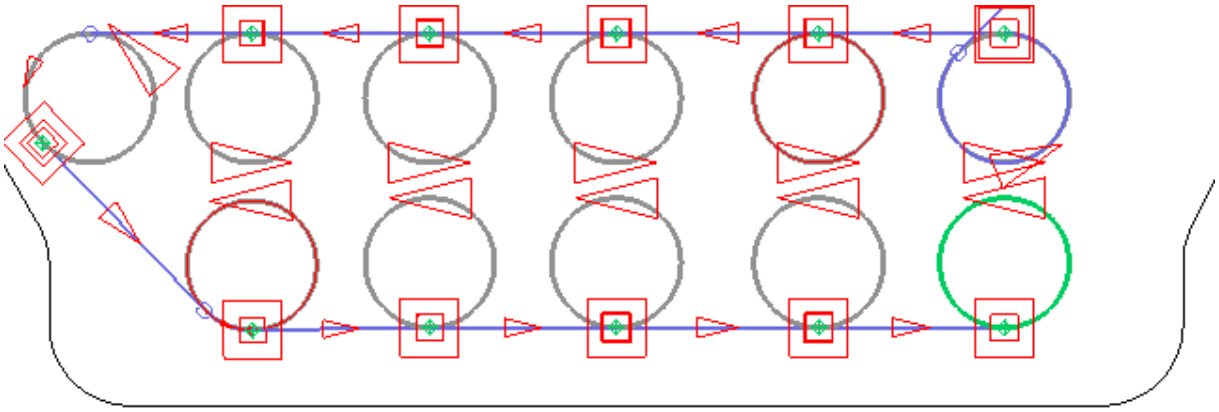
Manual mode:

- Click on a line or a full circle (exclude the arcs).



When you have selected a line, it is possible to indicate a tangence length link after cutting and before the next cutting, when there is a change in direction.

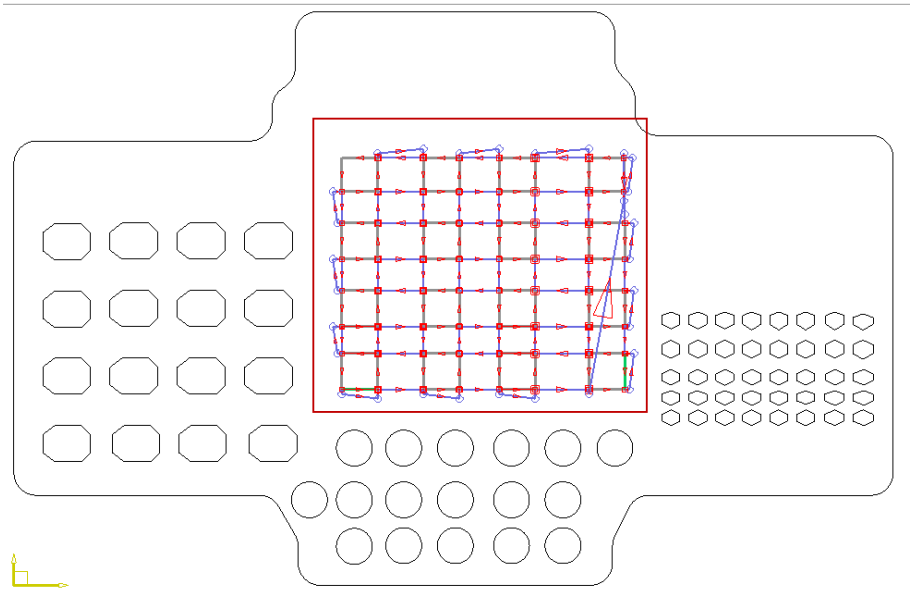
The **Invert** Yes/No button makes it possible to invert the direction of the tool path when the initial geometry does not turn in the desired direction.



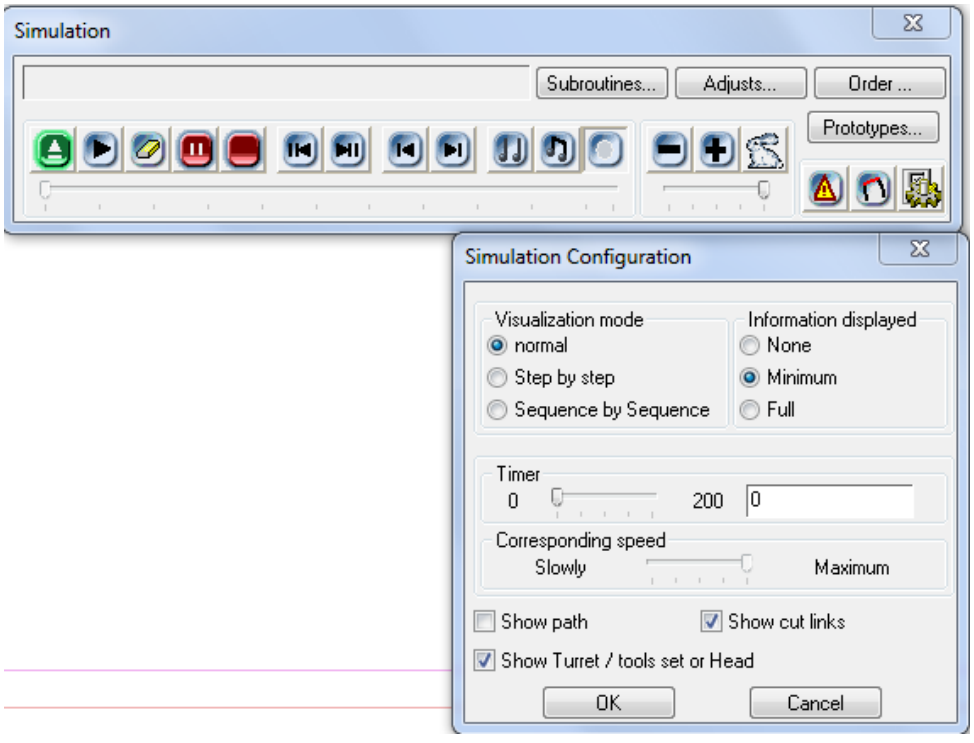
Example of circles

Automatic mode:

Automatic mode operates via a strict selection of lines or full circles.

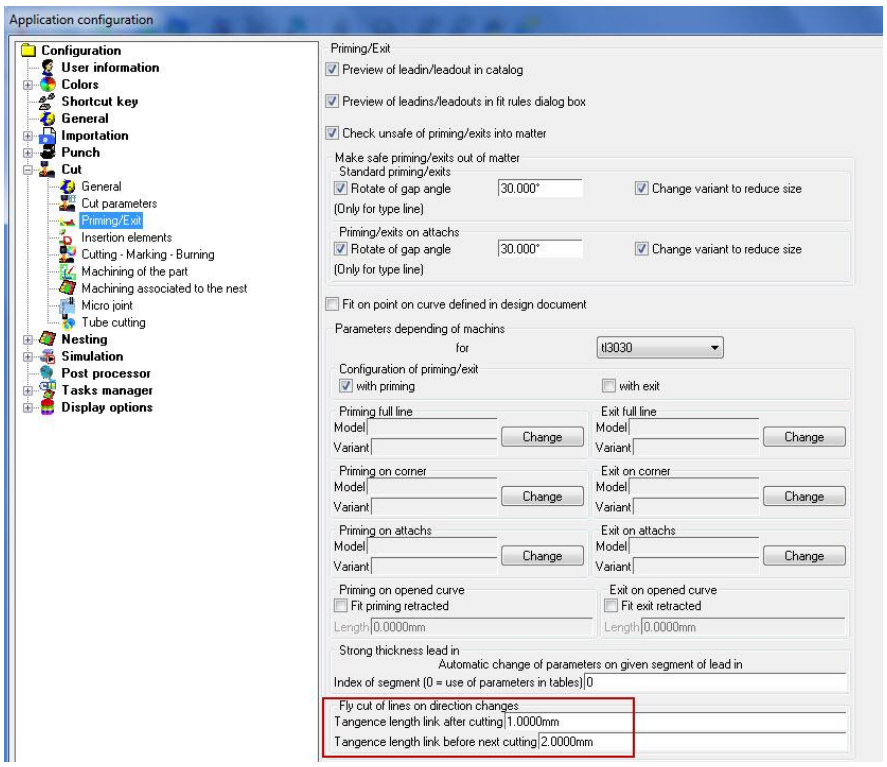


During simulation, possibility of not viewing the links between cuttings.



When the machine allows, post-processor installation is necessary.

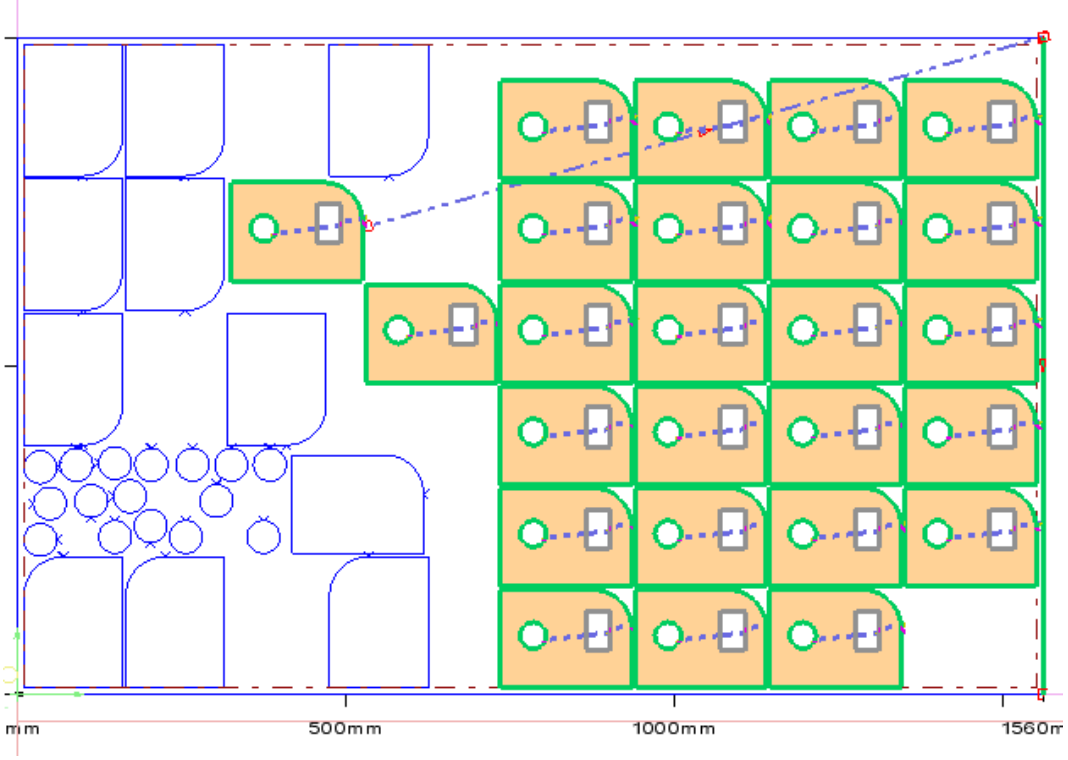
Settings in **Tools | Options**.



Lead in tracks –Saving Format as waste

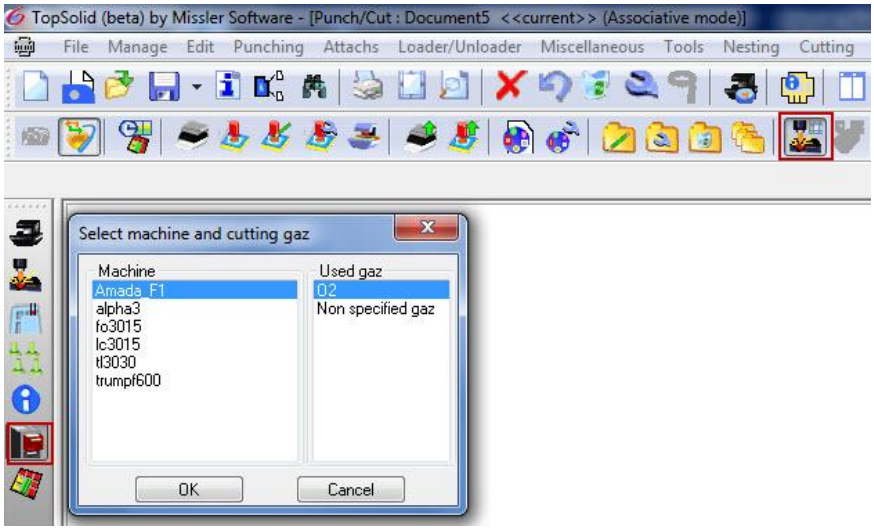
In the **Extract waste from format** function, possibility of saving the format as waste in order to reuse this format for a future nesting. In the previous versions, only the core/cavity of the external contour parts were memorized and it is possible, during the next nesting, that the parts were positioned where the previously positioned part lead ins were. This happened when the size of the lead in was greater than the gap between the parts.

Example of nesting to be saved as waste:



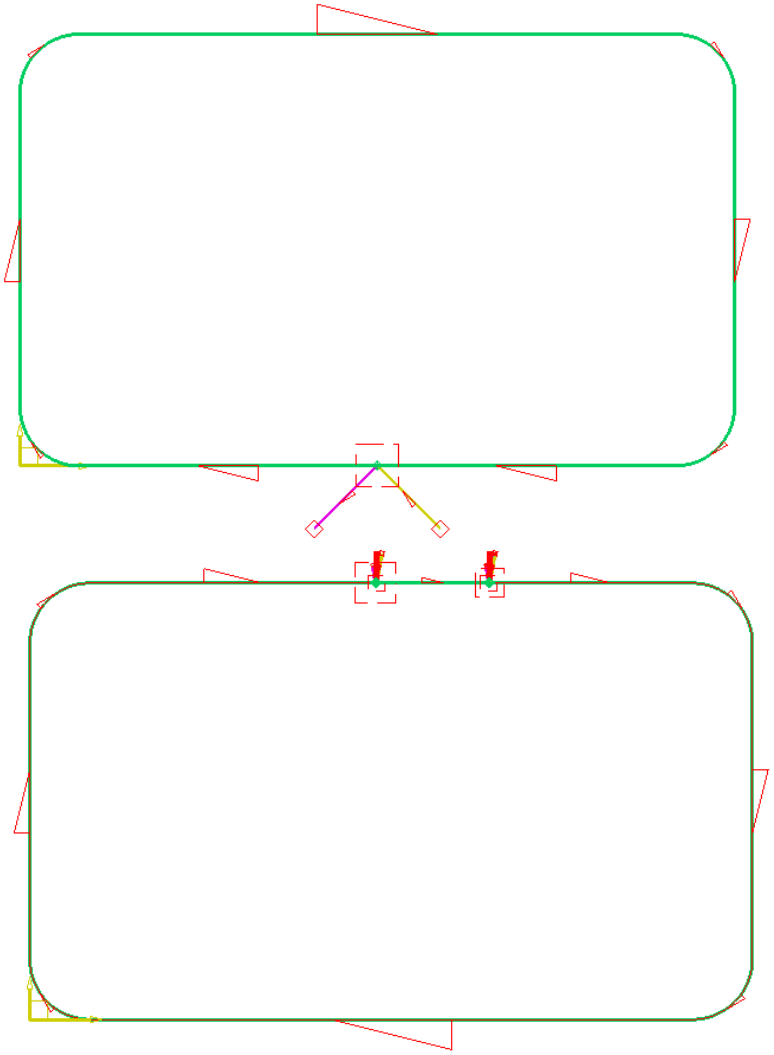
Cutting parameter access

When you have an empty part document or nesting document on the screen, it is possible to call up the **Cutting parameters management** function from the **Management** Menu. Select the machine as well as the cut gaz (associated with the machine) whose parameters you would like to input or edit.

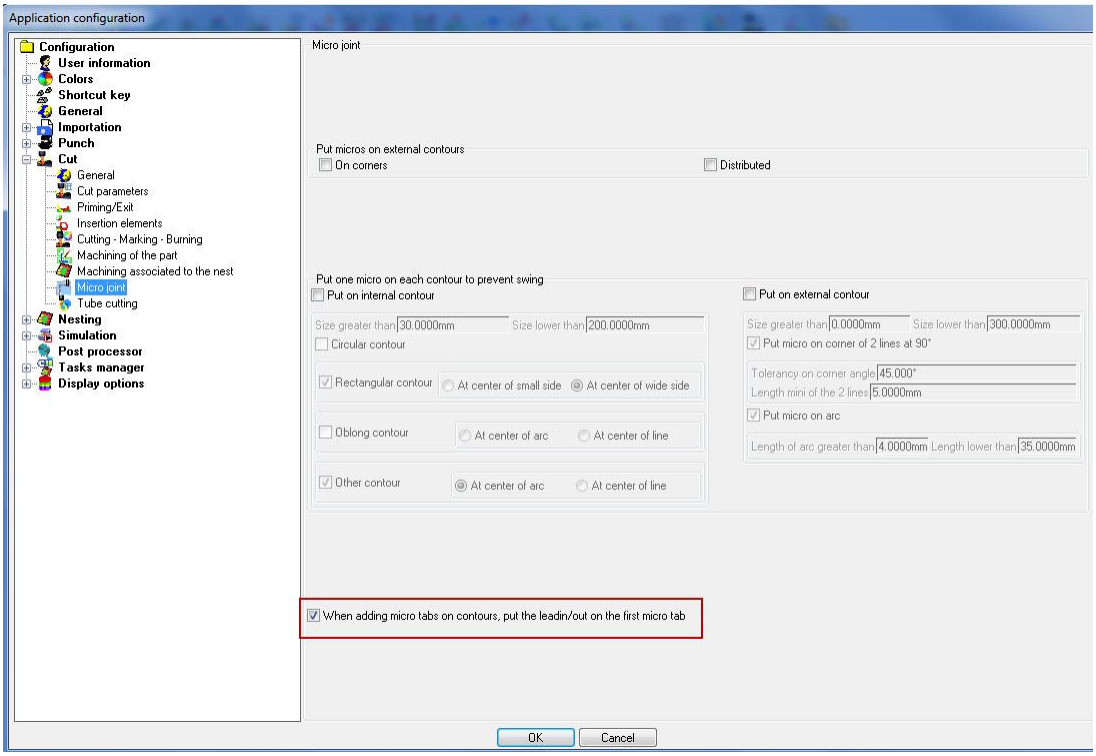


Move a lead in when adding micro joints on a cutting path

When you add micro joints on a previously machined contour, it is now possible to automatically reset the lead in/lead out on the first micro joint of the contour.



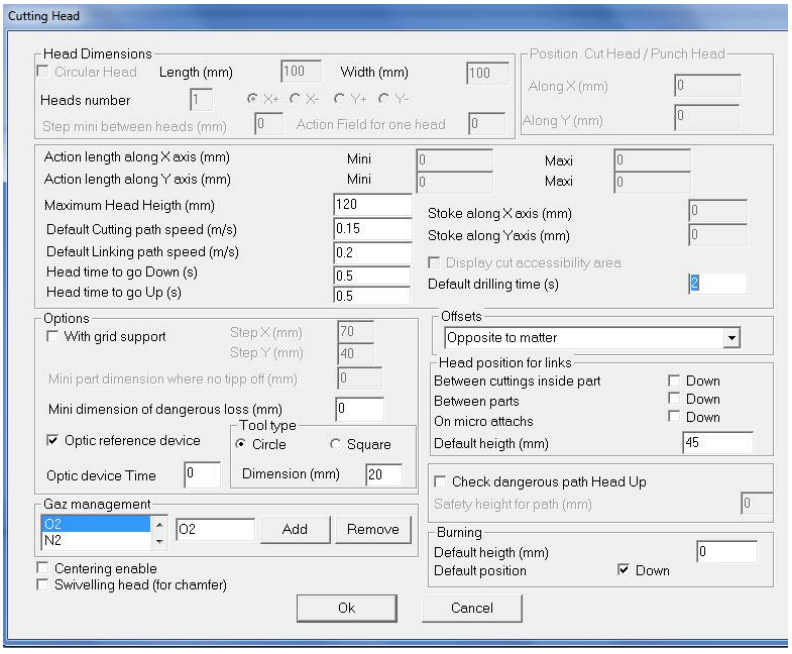
Configuration in **Tools | Options**.



Cutting head up-down times

In the aim of fine-tuning cutting machining times, possibility of configuring the time for the cutting head to go down and up.

Configuration is carried out in the **Management | Machines** menu in the **Cutting head** section.

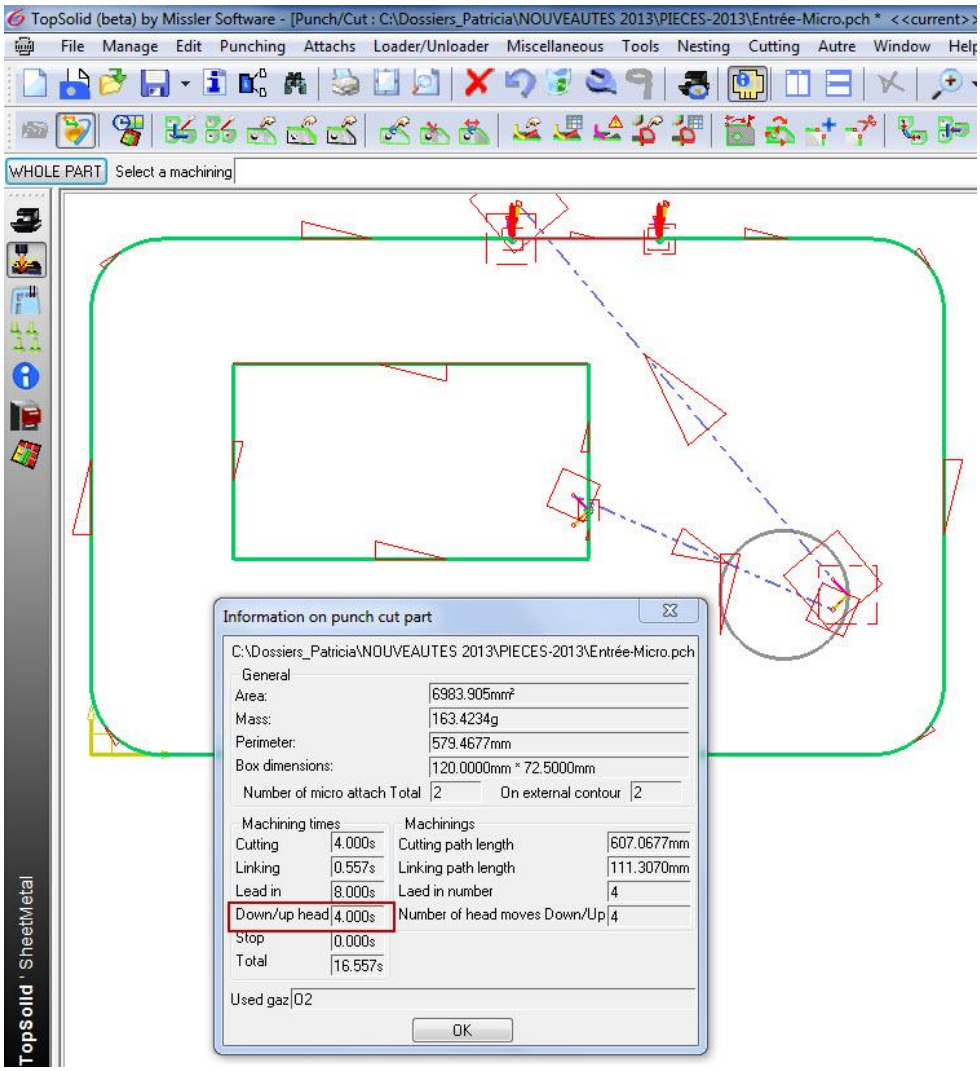


The time is displayed in an informational dialog box for the part or format.

It depends on the status of the links (head up or head down). If there is not a link between two cuts, the head is considered to go up and down between the two cuts.

If the times for the head to move up and down are set to 0. In the machine parameters, the time calculated in the information box on the part will be set to 0 and the number of head up/head down movements will not be entered.

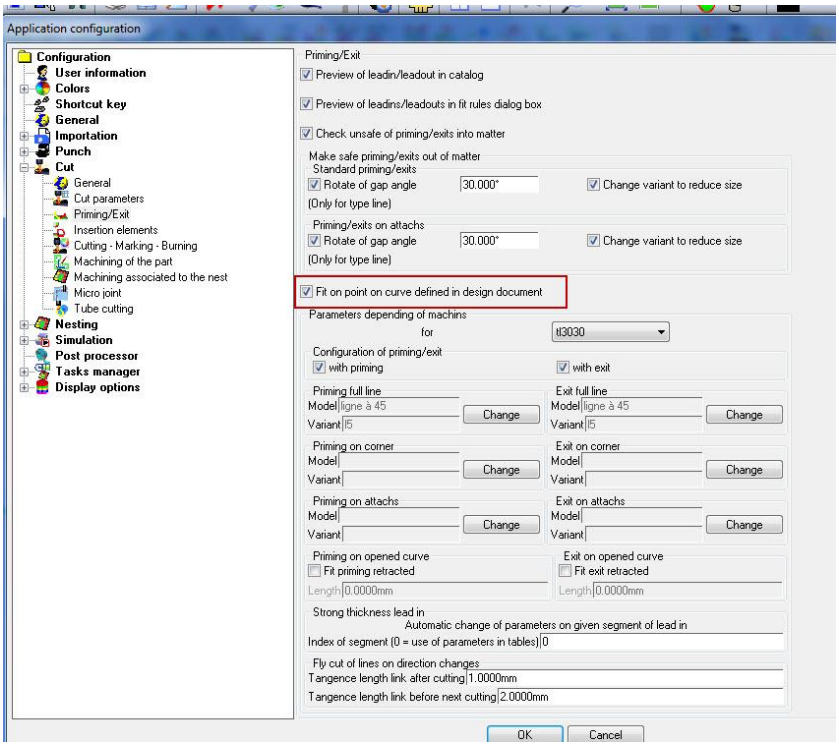
These times may appear in the DRAFT operator file of the nesting.



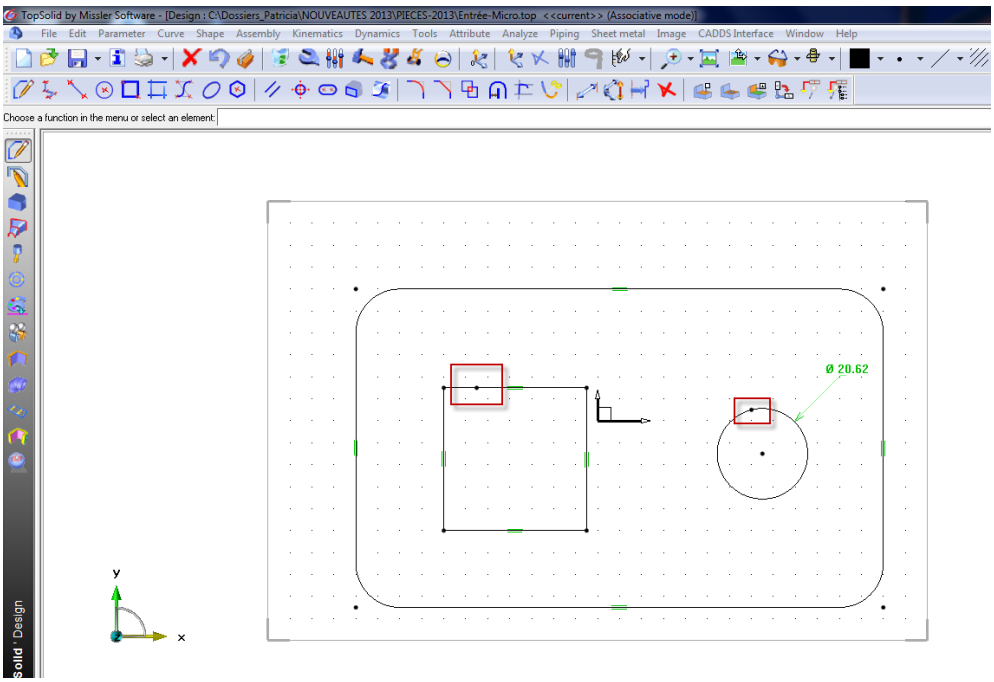
Fit on point on curve in the Design document

It is possible to set the fit point on a contour by creating a **point on curve defined in design document** (.top).

- To activate this possibility, look in **Tools | Options**.

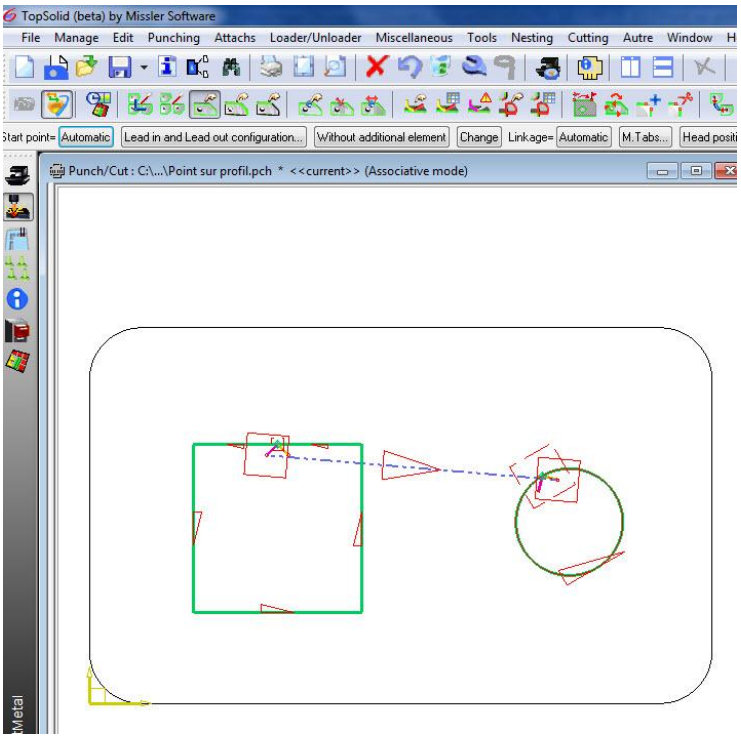


Implementation in the .TOP document:



- Use the **Tools | Point | Point on Curve** function and click on the curve.
  The color and type are inconsequential.

In the **Manual Contouring** function, using the automatic departure point, the system examines the point on curve in the .TOP document and, if it exists, positions the lead in in this location.



This principle applies by default to the **Automatic Contouring** and **Automatic contouring and links** functions.

If the machinings are carried out directly on the nesting, the fit point is recovered by transitivity.

When the lead in has been set, it will not move unless the point on curve is moved subsequently in the .TOP document However, if machining is carried out again, the fit point will take the new position.

Warning: if micro joints exist, by default the lead in will be set on the point corresponding to the point on curve in the .TOP document.

Taking into account of the exit in complex nestings

In complex nestings, if you request taking into account of cut allocations, the system now takes into account the point of laser entry and exit to avoid part collisions.

Punching and cutting

Display of the quantity of symetrized parts on a separate line

In the nesting document, beneath the format, are indicated the names of the parts in place with their quantities. Possibility of dissociating the parts that have a symetry in X and/or Y on a different line.

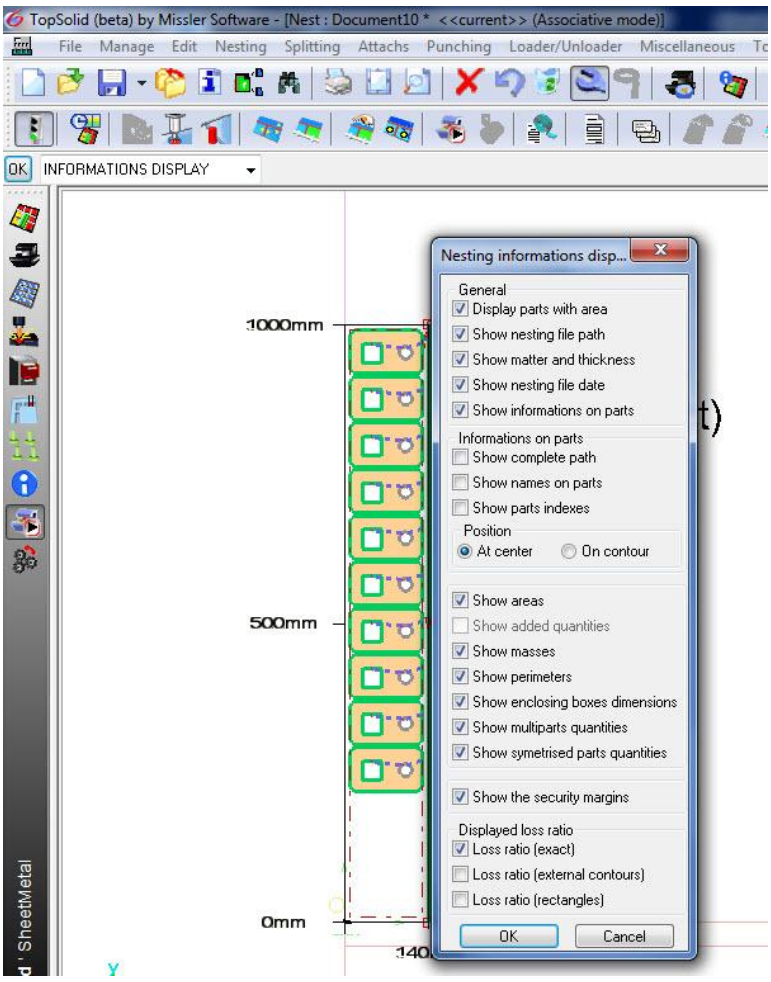
This option can be configured directly in the document display options.

- Use the wrench on the format and select **Display configuration**.
- Check the **Display quantities of symetrized parts** box.

This option is not compatible with the display of added quantities. It is therefore not available if the added quantities are requested for a given nesting.

Configuration possible from **Tools | Options.**

Tools | Options | Nesting | Nesting information display.

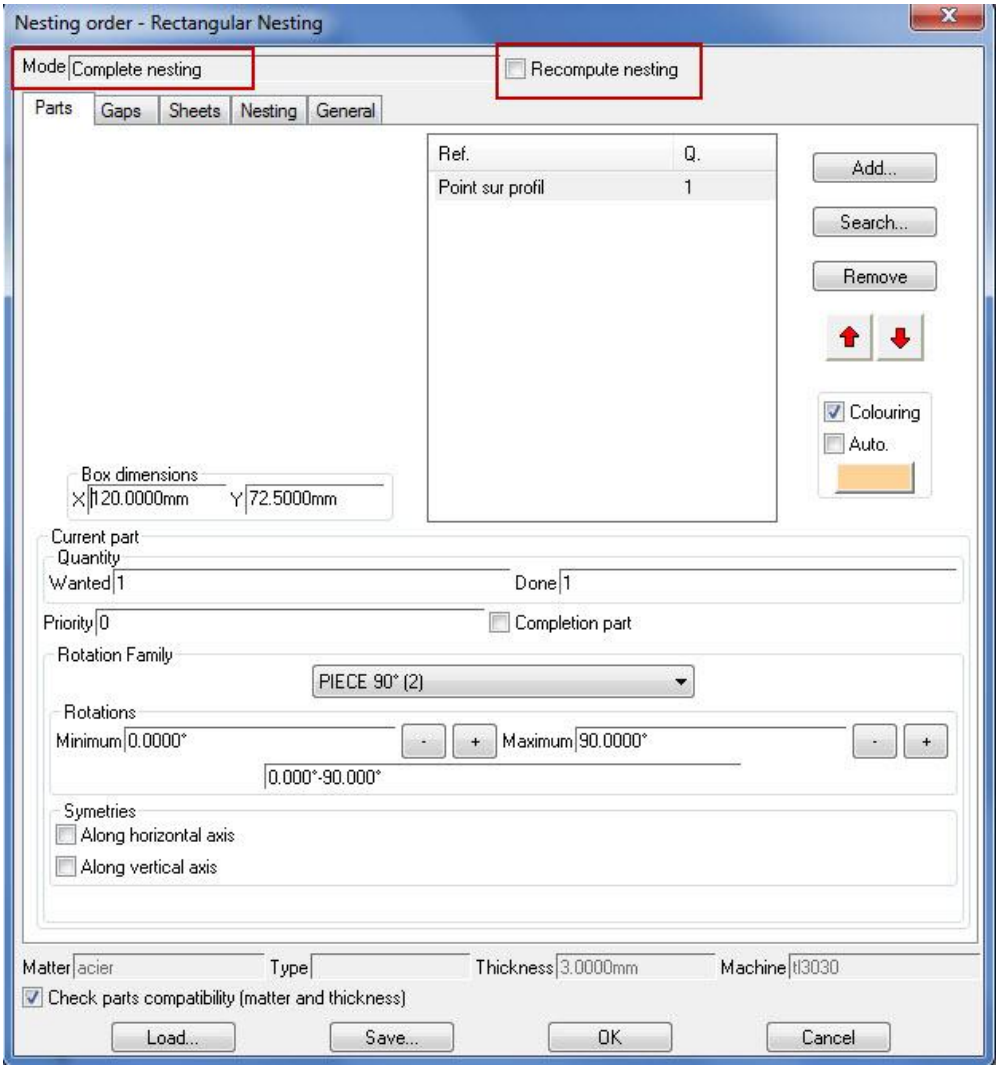


Copy Schedulings

The **Same machining as** function in **Multiple** mode now offers the option of copying schedulings included in the group of items whose machinings you wish to copy.

Complete Rectangular Nesting

Possibility of adding parts in the rectangular nesting by "completing" it without replaying the entire nesting.

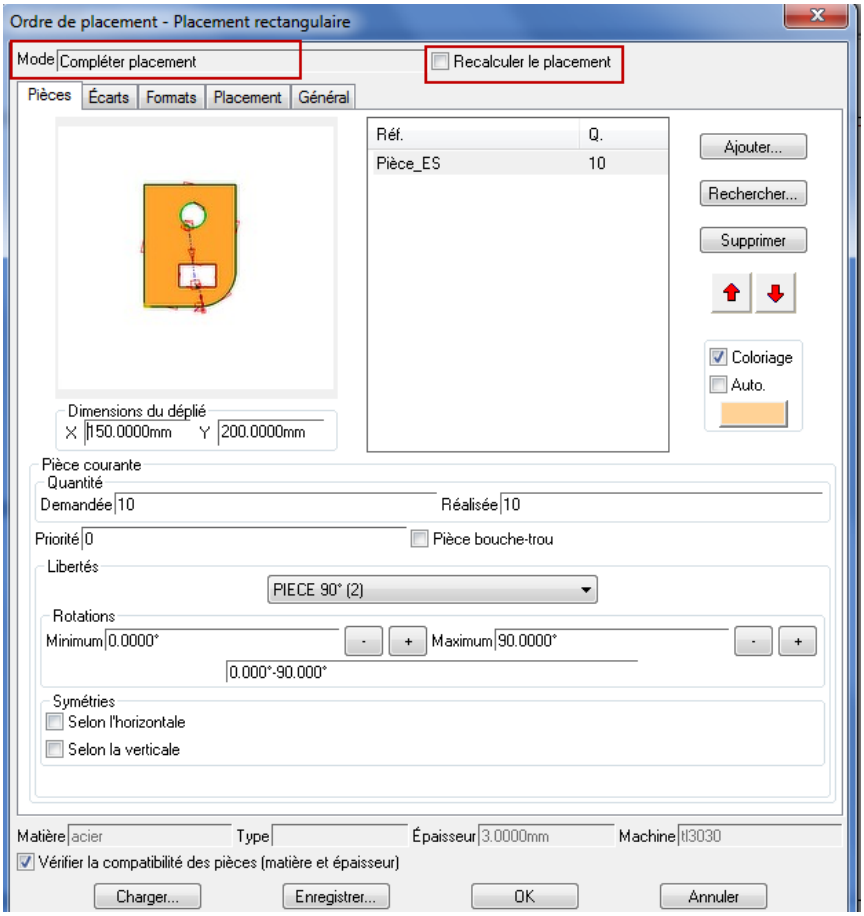


Tasks manager

In cluster format from the Manager

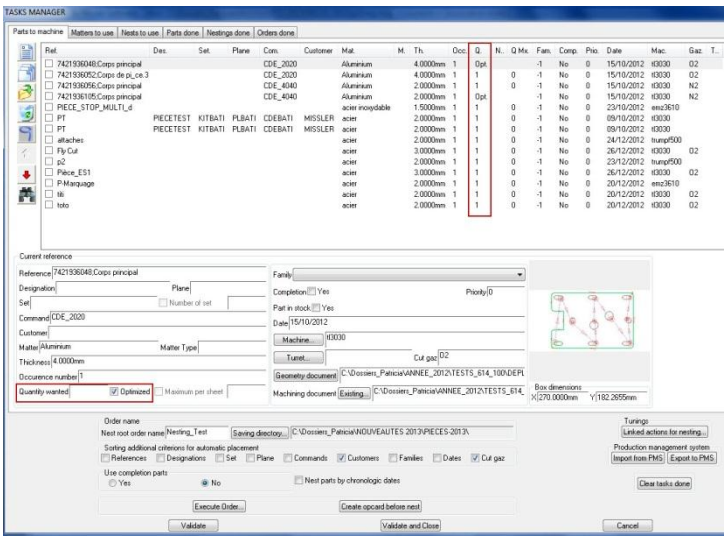
When you use a format in the Tasks manager, there is the possibility of creating a **Cluster** type sub-format from a part that appears in the Manager.

After the creation of the sub-format and its validation, the part quantity of the sub-format is subtracted from the Tasks manager.



Optimize Part Quantity

Possibility of making an order from the part manager in optimized quantity. The quantity is not entered and the nesting module will provide the quantity made on the requested format.

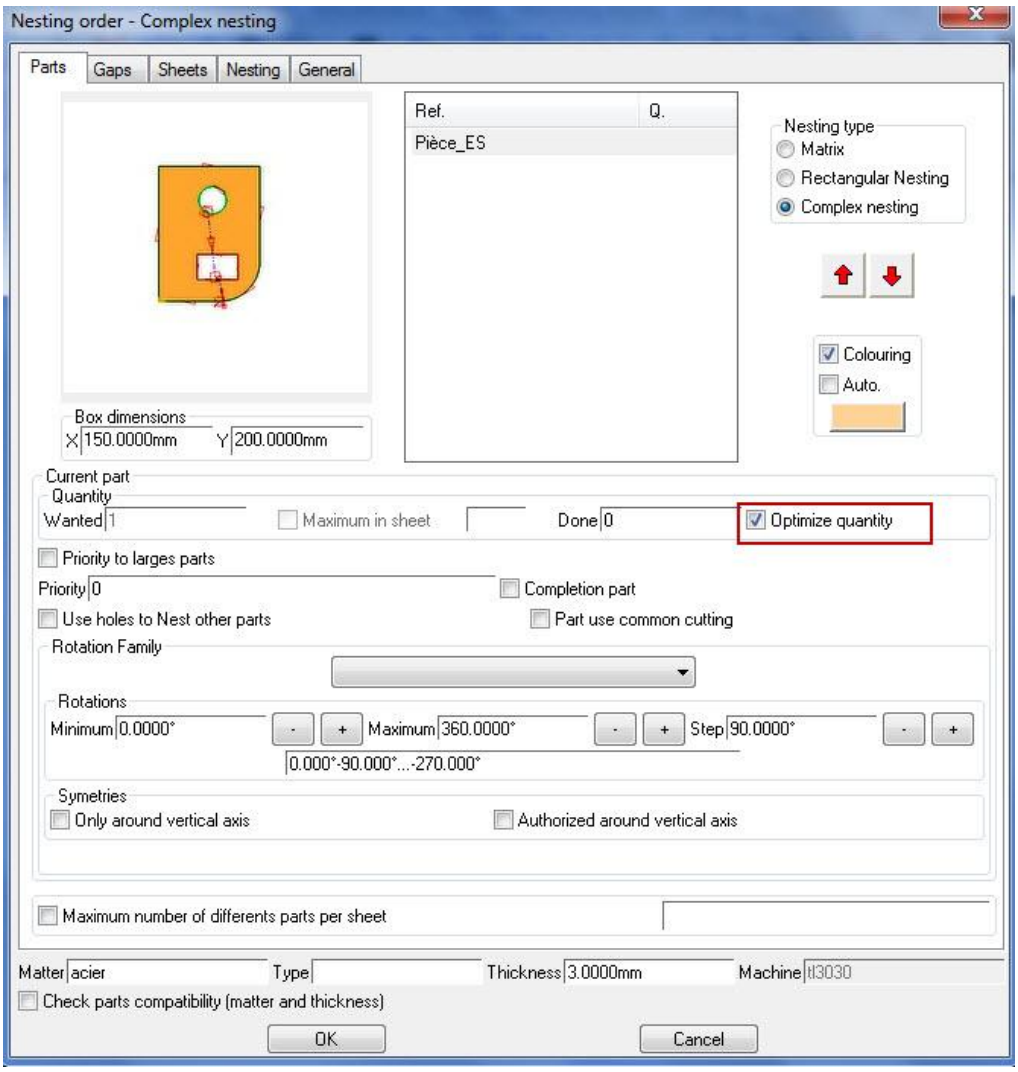


This gives access to the **Optimize quantity** box in the nesting order dialog box

Warning: Mixing parts on a single format in optimized quantity and for parts in set quantity is not possible.

A block is carried out before starting the orders.

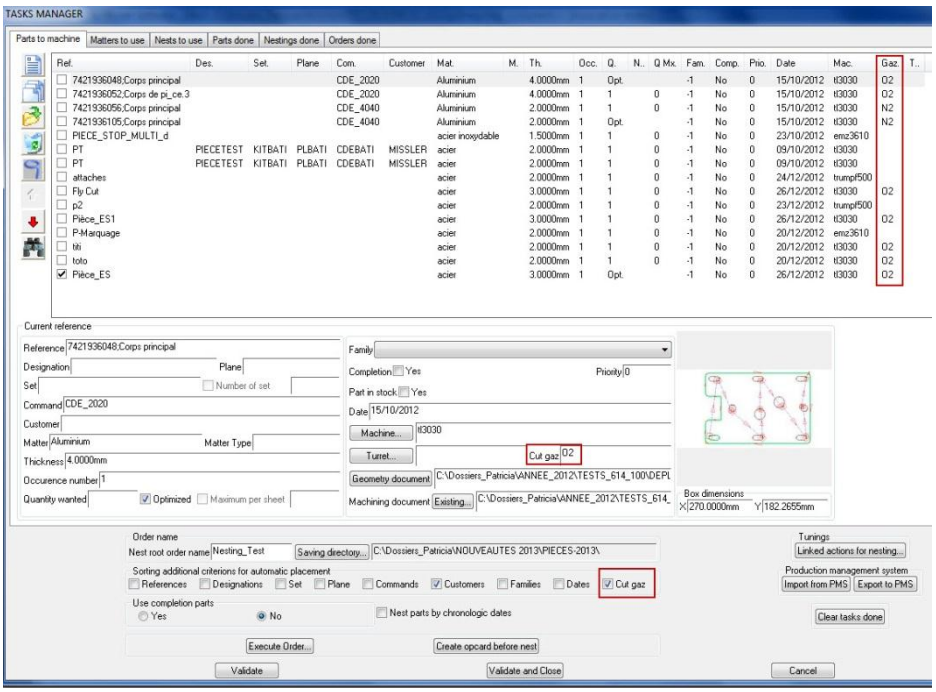
If you would like to launch several nestings at once with a part in optimized quantity for each one, it is necessary to check the additional grouping criterion box **References**.



Cut gaz grouping criterion

The tasks manager now manages cut gaz for cutting machines.

The **Cut Gaz** field, the column and the additional grouping criterion are available in the Manager dialog box.

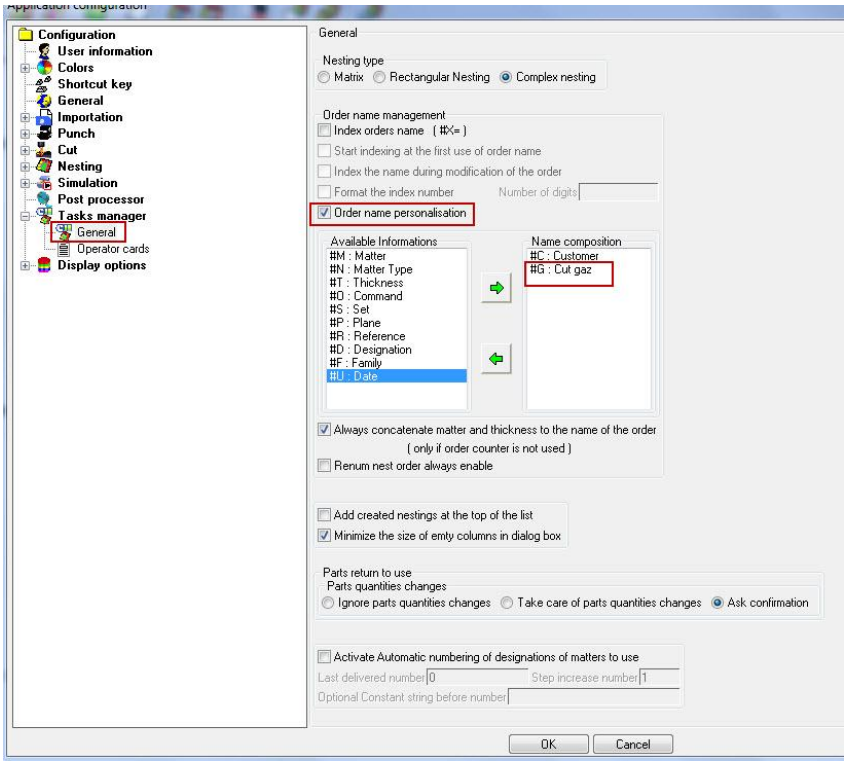


If the additional grouping criterion **Cut Gaz** is not activated, parts with different gases may be mixed.

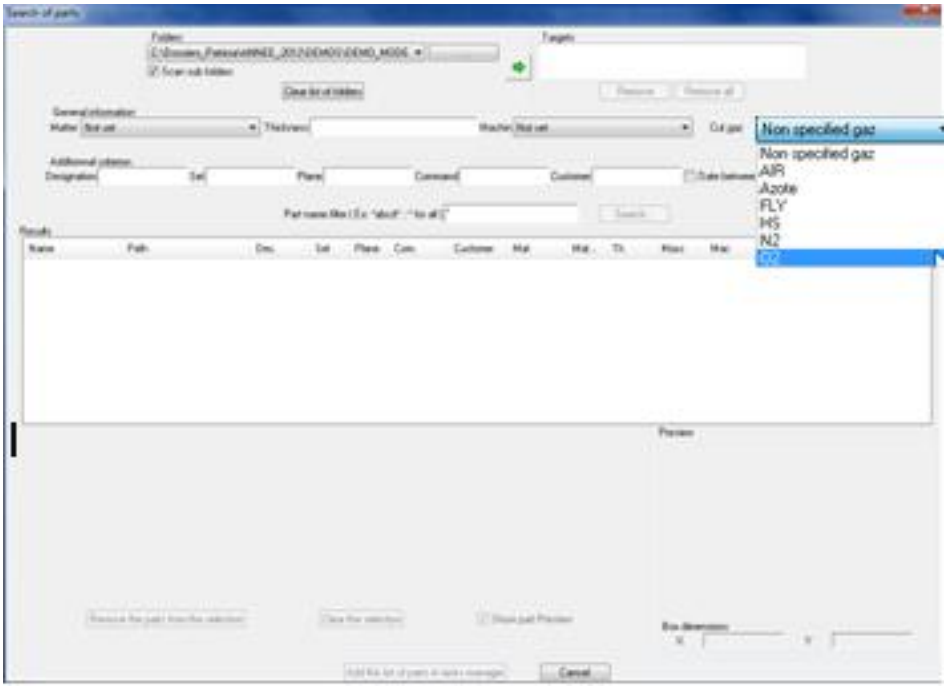
If the box is checked, orders will be distinguished.

To customize the cut gaz position in the order name, use **Tools | Options**.

The gas will be preceded by *# G=*.



The **Search of parts** function in the **File** menu includes the cut gaz filter.

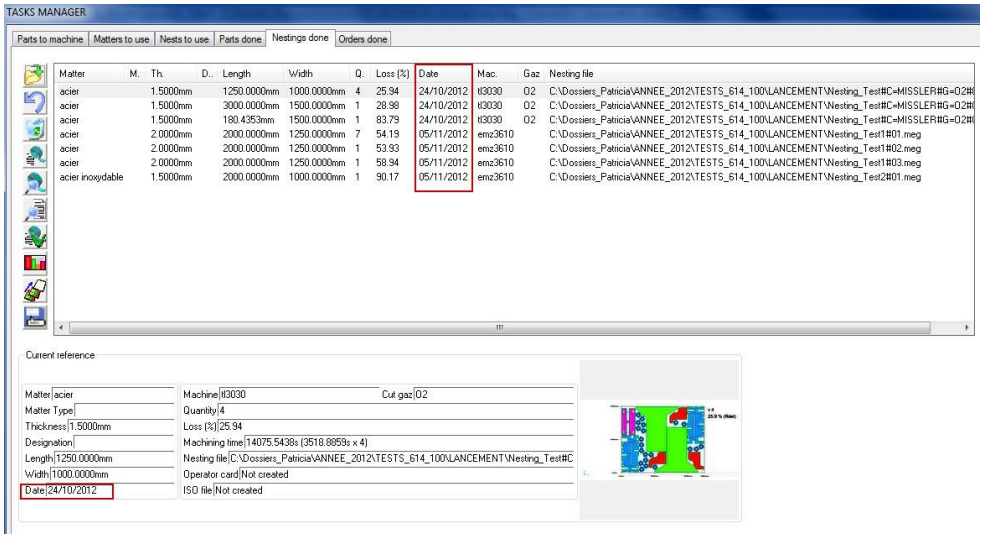


If no filter on the machine is active, all gases used on all machines are shown in the combo box. Otherwise the list of gases is limited to those of the selected machine.

When the cut gaz is modified in the PCH part and if the part is already present in the Manager with a different gas, the system suggests updating the gas in the Manager.

Date Column of the Tasks Manager

A new **Date** column has been added in the **Nestings done** tab.



Tasks Manager Backup

The tasks manager is archived as soon as it is saved. The backups are kept and there are three of them in the file: *$PUNCHDATA\Archives_TMan.*

In the event that the Manager is damaged, it is possible to restore a backup.

Example:

Tasksman.tmn#B=-1.bak	Archive the most recent
Tasksman.tmn#B=-3.bak	Archive the oldest

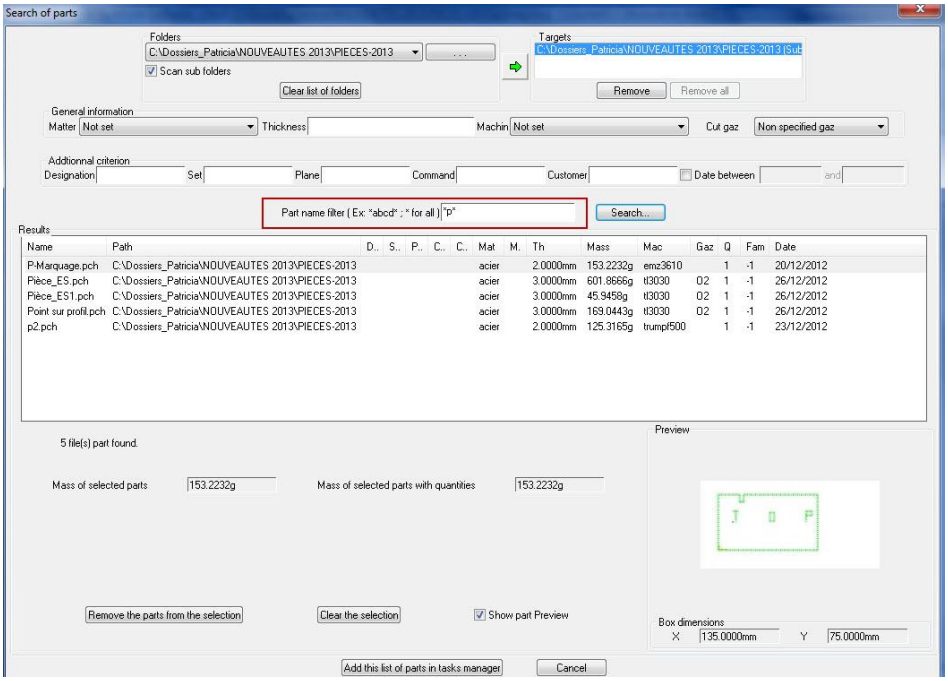
To restore a backup, copy the filen*$PUNCHDATA\Archives_TMan\tasksman.tmn#B=-1.bak* into *$PUNCHDATA\Archives_TMan\tasksman.tmn*.

This operation must be carried out when TopSolid is not running.

Additional criteria - Search of parts

A filter has been added to the **Search of parts** function to find parts by name.

This filter meets the Windows search concept by supporting the "*" character (e.g. *P*, by default you search for all parts which comes back to *).



Additional search criteria are also available.
The criteria that you would like to see added in the dialog box are defined in **Tools | Options**.